SHIPCONSTRUCTOR®

2013

Featuring: Database Driven Relational Object Model™ (DDROM™).

Copyright 2012 ShipConstructor Software Inc. All Rights Reserved.

Structure

www.ShipConstructor.com

© Copyright 2012 ShipConstructor Software Inc. – Feb. 1, 13

ShipConstructor 2012 Structure
Published 2013-02-01

Copyright
Copyright © 2012 ShipConstructor Software Inc.

Information in this ShipConstructor manual is the property of ShipConstructor Software Inc. No part of it can be reproduced, translated, resold, rented, adapted, modified, stored in a retrieval system or transmitted in any form or by any means, in whole or in part. All Rights Reserved.

Trademarks
ShipConstructor
AutoMagic
SmartParts
Database Driven Relational Object Model
DDROM

Are all registered trademarks of ShipConstructor Software Inc.

ShipConstructor Software Inc.
Suite 304
3960 Quadra Street
Victoria, BC
Canada V8X 4A3

Toll Free: 1-888-210-7420
Phone: 1-250-479-3638
Fax: 1-250-479-0868

Information: info@ShipConstructor.com
Support: support@ShipConstructor.com
Sales: SSI@ShipConstructor.com

Website: www.ShipConstructor.com

SHIPCONSTRUCTOR LICENSE AGREEMENT

BEFORE PROCEEDING WITH THE INSTALLATION, YOU MUST ACCEPT THE TERMS OF THIS AGREEMENT. INDICATE YOUR ACCEPTANCE OR REJECTION OF THIS AGREEMENT BY CLICKING ON THE APPROPRIATE BUTTON. IF YOU CLICK ON "REJECT," INSTALLATION WILL ABORT.

1. License Grant. ShipConstructor Software Inc., #304 – 3960 Quadra Street, Victoria, B.C. Canada, V8X 4A3 ("SSI") grants to the person accepting this Agreement (the "Licensee") a non-exclusive, non-transferable right to use (the "License") in object code form those program modules, application programming interface ("API"), any other materials provided by SSI under this Agreement, and all upgrades, revisions, fixes, updates or enhancements to any of the foregoing ("Licensed Materials") specified in the Licensee's purchase order or request ("Invoice) solely on the software and hardware listed in the Licensed Materials manual ("System Configuration").

2. Academic Institutions/Trial Versions.

   A. In the event that the Licensee qualifies as an academic institution user in accordance with SSI's specifications (an "Academic Licensee"), the Academic Licensee and its faculty, employees and students may use the Licensed Materials for the singular purpose of either teaching, training users or undertaking research provided that the Licensed Materials, and all copies of the Licensed Materials, remain at all times at the Academic Licensee's premises and the Licensed Materials are used for no other purpose than that set forth above. The above restrictions are in addition to the restrictions on use set out in Section 5 below.

   B. In the event that the Licensee receives a trial version of the Licensed Materials for evaluation purposes, the terms and conditions of this Agreement, excluding Sections 15-19, shall continue to apply subject to the following provisions:

      (a) the License pursuant to Section 1 above shall terminate at the end of the specified trial period;

      (b) the Licensee shall return the Hardware Key to SSI immediately upon expiry of the specified trial period and in any event within 28 days of the expiry of the specified trial period;

      (c) in the event that the Licensee does not return the Hardware Key in accordance with Section 2B.(b) above, SSI shall be entitled to invoice the Licensee for and the Licensee shall pay for the costs of the Hardware Key plus all shipping and handling expenses and SSI administrative charges; and

      (d) in the event that the Licensee elects to and does acquire a License, the terms and conditions of this Agreement, excluding Section 2B herein, shall continue on and apply.

3. Ownership. All rights, title and interests in and to the Licensed Materials and related documentation shall remain the sole property of SSI. Licensee shall not remove or alter any proprietary rights notices on the Licensed Materials and the documentation, and shall reproduce such notices on any copies that it makes. Licensee shall be liable for the security of the Licensed Materials and the documentation in its possession.

4. Expertise Required. Licensee is responsible for evaluating whether the Licensed Materials meets Licensee's requirements, and for operating the Licensed Materials and the results obtained. The Licensed Materials are intended for ship modeling and construction purposes only, and must be used by a person who has expertise and knowledge in this field. The Licensed Materials requires independent confirmation of the reliability and accuracy of all designs, drawings and other Licensed Materials output. An SSI representative may be made available under a separate consulting agreement, at the Licensee's request to provide training and consultation on the operation or integration of licensed materials.

5. Limitations on Use.

   Licensee shall:

   (a) not make more copies of the Licensed Materials than are necessary for the Licensee's installation of the Licensed Materials and shall only create backup copies for archival or emergency restart purposes;

   (b) maintain a log of the number of and location of all originals and copies of the Licensed Materials;

   (c) include SSI's copyright, trademark and proprietary notices on any complete or partial copies of the Licensed Materials in the same form and location as the notice on any original work;

   (d) not attempt to defeat any copy protection;

   (e) not modify, any documentation, including any user manuals;

   (f) not modify, translate, reverse engineer, decompile or disassemble the Licensed Materials;

   (g) not sublicense, transfer, assign, sell, loan, rent or lease the Licensed Materials other than as permitted in this Agreement;

   (h) use the Licensed Materials for its own internal use only;

   (i) not permit any third party to use the Licensed Materials; and

   (j) thoroughly test any and all custom interfaces in accordance with general engineering principles.

6. Delivery and Installation. All Licensed Materials will be delivered in an electronic format by media or method as SSI may elect and will be sent to the Licensee's

designated email address or shipping address as specified in the Invoice. Licensee agrees to be responsible for installation of the Licensed Materials.

7. Term of License. The License term commences on the delivery of the Licensed Materials to the Licensee, and, subject to Section 2B above, is either perpetual if so requested on the Order, or on a month to month rental or lease basis. If Licensee chooses a lease option the license converts to a perpetual term on Licensee's payment of the balance of the perpetual License fee (prior monthly payments receiving 80% credit). All Licenses are subject to termination in accordance with this Agreement.

8. System Configuration. Operation of the Licensed Materials requires use of the specified System Configuration, which Licensee shall acquire and implement. SSI shall not be responsible for any operational problems caused by the System Configuration.

9. Security. The Licensed Materials includes security elements which support the detection of unauthorized use or copying of the software and which may (a)report such unauthorized use or copying to the Licensee, and (b)if applicable based on Licensee's configuration, may report back specific user information such as User name and email address.

10. Hardware Keys. Licensed Materials use requires "Hardware Keys" supplied by SSI, which can be used only at the site(s) authorized by SSI. In the event of a failure of the Licensee's System Configuration, the Licensee may upon advising SSI use the Hardware Keys and Licensed Materials on another system and/or location.

11. License Fees. Licensee shall pay to SSI the License fees applicable for the Licensed Materials as set out in and in accordance with SSI's Invoice.

12. Services. Support services after the Warranty Period (as defined in Section 15 below) are provided by SSI under the terms of the SSI Subscription Agreement. Installation, consulting, training and implementation services, if requested by the Licensee, shall be provided by separate agreement and at an additional charge.

13. Taxes. All amounts payable by Licensee to SSI are exclusive of all commodity taxes, including but not limited to applicable sales, use, value added, custom duties, excise taxes and other similar government charges, all of which will be paid by Licensee. If Licensee is required by law to withhold any taxes, then Licensee shall pay SSI a gross amount of money such that the net amount received by SSI after deducting or withholding the required taxes is equal to the amount of the fee originally charged by SSI.

14. Interest Charges. If any amount payable under this Agreement is not paid within 30 days of becoming due, SSI shall have the right to impose a charge of 2% per month (24% annually) on the unpaid balance of the amount, from the due date until the date of receipt of all amounts in arrears including interest.

15. Purchase Orders. Any purchase order (an "Order") delivered by Licensee shall at all times be deemed to incorporate this Agreement by reference and shall be subject to the applicable provisions of this Agreement. Any provisions of an Order shall not apply and shall not be binding upon SSI unless they relate to information which was requested by SSI. In the event of a conflict or an inconsistency between the provisions of an Order and the terms and conditions of this Agreement, this Agreement shall govern and supersede to the extent of such conflict or inconsistency.

16. Limited Warranty. SSI warrants that during a period of 90 days from the date of delivery of the Licensed Materials to Licensee (the "Warranty Period"), the Licensed Materials will perform substantially in accordance with the Licensed Materials documentation specifications, when used in accordance with this Agreement on a properly operating System Configuration. SSI's sole obligation under this Warranty, and Licensee's exclusive remedy, shall be to use reasonable commercial efforts to correct Errors (a bug, defect or other problem incurred by a user in operating the Software that prevents the Software from performing in a manner consistent with the applicable specifications set out in the User Manual) that the Licensee identifies to SSI through fixes or workarounds free of charge. If SSI determines that it is unable to make the Licensed Materials perform substantially as warranted, Licensee may terminate the License and receive a refund of a portion of the License Fees paid to date.

17. WARRANTY EXCLUSIONS. THE LIMITED WARRANTY CONTAINED IN SECTION 15 IS IN LIEU OF ALL OTHER WARRANTIES, EXPRESS OR IMPLIED. ALL OTHER CONDITIONS, WARRANTIES, AND REPRESENTATIONS, EITHER EXPRESS OR IMPLIED, ARE EXCLUDED, INCLUDING BUT NOT LIMITED TO CONDITIONS, REPRESENTATIONS AND WARRANTIES RELATING TO MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. SSI DOES NOT WARRANT THAT THE LICENSED MATERIALS ARE COMPLETELY ERROR FREE OR THAT ITS OPERATION WILL BE CONTINUOUS AND UNINTERRUPTED.

18. Maintenance Services. Licensee may elect, to obtain maintenance, support and upgrade services from SSI in accordance with and subject to the terms of SSI's standard Subscription Agreement.

19. Loss of Data. SSI shall not be responsible for any loss of or damage to files or data caused by the Licensed Materials, or be required to restore or rebuild files or data. Licensee shall implementing adequate backup procedures to avoid any loss of files and data.

20. Modifications. SSI may, from time to time, provide the Licensee with revisions to the Licensed Materials (the "Revised Licensed Materials"). The Licensee shall test any external applications using the revised API before implementing the new version. While it is SSI's intention that the Revised Licensed Materials shall be backward-compatible with the immediately prior version of the Licensed Materials, SSI does not guarantee or warrant that this shall be so, and SSI shall have no liability whatsoever to the Licensee for any failure of the Revised Licensed Materials to be backward compatible with any prior version of the Licensed Materials. Modifications requested by the Licensee shall be subject to prior written agreement as to scope and fees payable. Ownership of all Licensed Materials modifications shall vest in SSI. SSI does not warrant, guarantee or otherwise commit to supporting Licensed Materials that has been superseded by Revised Licensed Materials.

21. Confidential Information. Each party will not use the confidential information of the other party for any purpose except for the purpose described in this Agreement, and shall not disclose it to any other person except on a confidential basis to its employees and representatives who have a need-to-know the confidential information for such purposes. This Section 23 shall not apply to confidential information which (a) is or has become readily available to the public in the same form other than by an act or omission of the receiving party, (b) was lawfully obtained in the same form by the receiving party from a third party not under an obligation of confidence to the disclosing party, (c) was in the receiving party's possession in the same and material form prior to its receipt from the disclosing party and did not otherwise originate from the disclosing party, or (d) is required to be disclosed by operation of law.

22. Audit Rights. Upon reasonable notice by SSI, which shall be delivered on no more than an (annual?) basis, Licensee shall provide a signed statement verifying its compliance with the terms of this Agreement. SSI shall also have the right, upon reasonable notice and no more than on an annual basis, to inspect Licensee's facilities to verify Licensee's compliance with such terms. Any such inspection or audit shall be conducted either by SSI or by representatives authorized by SSI to complete the inspection. If such inspections or audits disclose that the Licensee has installed, accessed or permitted access to the Licensed Materials in a manner that is not permitted under this Agreement, then Licensee shall be liable to pay for any unpaid license fees as well as the reasonable costs of the audit.

23. Termination. This Agreement may be terminated by either party, immediately by written notice, if the other party commits a breach of any material provision of this Agreement, including a failure to make payment when due, and fails to correct or rectify such breach within 30 days of receipt of the notice requesting it to do so. SSI shall be entitled to place time-lock devices and other disabling features in the Licensed Materials that become effective in the event that the Licensee has failed to comply with its payment obligations hereunder and as set out in SSI Invoices.

24. Effect of Termination. Upon termination of this Agreement Licensee shall immediately cease using the Licensed Materials, and within 14 days of termination return all Hardware Keys to SSI.

25. CONSEQUENTIAL DAMAGES. IN NO EVENT SHALL SSI BE LIABLE FOR ANY LOSS OF DATA OR PROFITS, ECONOMIC LOSS OR SPECIAL, INDIRECT, INCIDENTAL, CONSEQUENTIAL OR PUNITIVE DAMAGES WITH RESPECT TO THIS AGREEMENT OR THE LICENSED MATERIALS, HOWEVER CAUSED, EVEN IF SSI HAD OR SHOULD HAVE HAD ANY KNOWLEDGE OF THE POSSIBILITY OF SUCH DAMAGES.

26. DAMAGES LIMITATION. THE MAXIMUM LIABILITY OF SSI FOR ALL CLAIMS AND DAMAGES OF ANY KIND, WHETHER FOR FUNDAMENTAL BREACH OR ANY OTHER CAUSE UNDER THIS AGREEMENT, SHALL BE LIMITED IN THE AGGREGATE TO THE TOTAL OF ALL FEES PAID BY LICENSEE.

27. LIMITATION OF NON-APPLICABILITY. IN SOME JURISDICTIONS THE EXCLUSION OR LIMITATION OF WARRANTIES OR LIABILITY MAY NOT BE APPLICABLE, AND IN SUCH JURISDICTIONS SSI HEREBY LIMITS ITS LIABILITY TO THE FULLEST EXTENT PERMITTED BY LAW.

28. Applicable Law. This Agreement shall be subject to and construed in accordance with the laws of the Province of British Columbia, Canada, excluding its conflict of laws rules and the application of the UN Convention on Contracts for the International Sale of Goods.

29. References. SSI shall be allowed to incorporate Licensee's name in SSI's customer reference list and to use it for marketing.

30. Dispute. If any dispute arises under this Agreement, a good faith attempt to resolve the dispute will be made by senior management of both parties at a mutually agreeable site and time. If the parties are unable to reach agreement within 30 days after a request for such meeting, the dispute shall be referred to arbitration in English, before one arbitrator in Victoria, British Columbia, Canada, in accordance with the Commercial Arbitration Act of the British Columbia.

31. Entire Agreement. This Agreement contains the entire agreement between the parties and shall supersede all prior discussions and agreements between the parties regarding its subject matter.

32. Amendment. Any amendment of this Agreement must be in writing and signed by duly authorized representatives of the parties.

33. Waiver. The waiver by any party of a breach by the other party of this Agreement shall not be construed as a waiver by such party of any succeeding breach by the other party of the same or another provision.

34. Assignments. Licensee may not assign or transfer the License or Licensee's rights or obligations under this Agreement without SSI's prior written consent, and any such assignment or transfer without consent shall be null and void. A transfer of all or substantially all of the voting stock of the Licensee shall constitute a transfer for these purposes and shall be subject to SSI's prior written consent.

35. Successors and Assigns. This Agreement will bind and enure to the benefit of the parties and their respective successors and permitted assigns.

36. Severability. In the event that any provision of this Agreement is declared invalid, illegal or unenforceable by a court having jurisdiction, then the remaining provisions shall continue in full force and effect.

37. Force Majeure. Except as related to Licensee's obligation to make payments to SSI, neither party shall be liable for delays or non-performance if such delays or non-performance are beyond such party's reasonable control. A delayed party shall promptly notify the other party in writing stating the cause of the delay and its expected duration and shall use commercially reasonable efforts to remedy a delay or non-performance as soon as reasonably possible.

38. Survival. The provisions of Sections 3, 5, 11, 13, 14, 17 and 20-30 shall survive the expiry or termination of this Agreement.

39. Language. It is the express will of the parties that this Agreement and related documents have been prepared in English. C'est la volonté expresse des parties que la présente Convention ainsi que les documents qui s'y rattachent soient rédiges en anglais.

#363338
28/05/2010

Contents

ShipConstructor 1

Getting Started 7

Start ShipConstructor 7
Check Your System Font 7
Open a Project 11
Get Licenses 12
Re-Login to ShipConstructor 13
Reload a Project Database 13
Change Your Password 13

Drawings 14

Open a Drawing 14
Create a Drawing 15
Lock or Unlock a Drawing 15
Rename a Drawing 15
Delete a Drawing 16
Save a Drawing 16
Closing a Drawing 16
Set Up Composite Drawings 16
Insert (Link) a Drawing into Another Drawing 17
Configuring Viewport Dependant Display Options 18

Exporting ShipConstructor Drawings 19

Export To Dwg 20
Export Drawings 22
Object Enabler 23
Views 24
Windows and Editors 24

Structure 25

Structural Design Workflow 25

Structure Library Setup 27

Overview 27
Supplied Libraries 27

Catalog Setup Workflow 27

Project Settings 27
Set Project Settings 27
Import Project Settings 28
Export Project Settings 28

User Permissions 28

Materials 29
Create a Material 29
Create a Grade 30
Rename a Material or Grade 30
Import Materials and Grades 30
Export Materials and Grades 30
Delete a Material or Grade 31
View Material Usage 31

Manufacturers 32
Create a Manufacturer 32
View Manufacturers Usage 32

Finishes 33
Create a Finish Type 33
Create a Finish 33
Rename a Finish or Finish Type 34
Import Finishes and Finish Types 34
Export Finishes and Finish Types 34
Delete a Finish or Finish Type 34
View Finishes Usage 34

Insulation 35

Text Styles 35
Create a Text Style 35
Delete a Text Style 35

Accessory Packages 35

Naming Conventions 35
Naming Conventions Window 36
Activate a Naming Convention 40
Rename a Naming Convention 40
Delete a Naming Convention 40
Managing Aliases 40
Generating Names in Groups 41

Naming Convention Concepts 42

Database Elements 43

Label Styles 44
Add a Label Style 45

Dimension Styles 45
Add a Dimension Style 45

Bill of Materials 45

BOM Functionality Overview 45

BOM Definitions Manager 47

Predefined Assembly Format 49

User-Defined Attributes 49
Create a User-Defined Attribute 49
Assign a User-Defined Attribute to a Part or Stock 50
Default Values on Required User-Defined Attributes 51
Import a User-Defined Attribute 51

Stock Catalog 51
Create a Stock 51
Edit a Stock 53
Create a Profile Stock Shape 54
Edit a Profile Stock Shape 66
Create a Cutout Type 66
Create a Cutout 67
View Cutout Usage 68
Import Stocks 69
Export Stocks 69
View Stock Usage 69

Piecemark Styles 70
Create a Piecemark Style 70
Edit Piecemark Style 71
Activate a Piecemark Style 71
View Piecemark Usage 72

Orientation Icon 72

Markline Styles 73
Create a Markline Style 73
View Markline Style Usage 74
Set Up Markline Symbols 74
Assign Markline Styles 77

Flange Catalog 79
Create a Flange Standard 79
Edit a Flange 80
Delete a Flange 80
Import Flanges 80
Export Flanges 80
View Flange Usage 81

Corner Treatments 81
Create a Corner Treatment 81
Delete a Corner Treatment 82
View Corner Treatment Usage 82

Plate Green Standards 83
Create a Plate Green Standard 83
View Plate Green Standard Usage 84

Bevel Standards 85

Create a Bevel Standard ... 85
View Bevel Standard Usage ... 86

Automatic Bevels ... 88

NC Machines ... 90
Create an NC Machine ... 90
Edit an NC Machine ... 90
Delete an NC Machine ... 90
Import NC Machines ... 91
Export NC Machines ... 91
View NC Machine Usage ... 91

Endcuts ... 92
Create an Endcut ... 92
Delete an Endcut ... 94
Import Endcuts ... 94
Export Endcuts ... 94
View Endcut Usage ... 94

Profile Green Standards ... 95
Create a Profile Green Standard ... 95
View Profile Green Standard Usage ... 96

Standard Parts Definition ... 97
Set Up Standard Part Types ... 97
Create a Standard Part ... 98
Edit a Standard Part ... 99
Rename a Standard Part ... 100
View Standard Part Usage ... 100

Dynamic Marking Block Standards ... 101
Set Up Dynamic Marking Block Types ... 101
Create a Dynamic Marking Block Standard ... 102
Delete a Dynamic Marking Block Standard ... 102
Edit a Dynamic Marking Block Standard ... 102
Import Dynamic Marking Block Standards ... 103
Export Dynamic Marking Block Standards ... 103
View Dynamic Marking Block Standard Usage ... 103
Preview Dynamic Marking Block Standard ... 103

Drawing Templates ... 104

Insert Empty BOM Wizard ... 105

Structure Setup ... 107

Import a Hull Model ... 107
Use a Hull Model from the Hull Module ... 107

Unit Setup ... 107
Create Units ... 107
Delete Units ... 108

Product Hierarchies ... 108

Setting Up Multiple Product Hierarchies ... 109
Copy Product Hierarchy ... 111
Export a Product Hierarchy to another ShipConstructor project ... 112
Import a Product Hierarchy from another ShipConstructor project ... 112
Set Up Product Hierarchy Drawings ... 113
Set Up Assembly Levels ... 116
Set Up User-Defined Attributes ... 117
Set Up Assemblies ... 119
Assign Parts, Spools, or Assemblies to Assemblies ... 122
Find, Zoom, Remove, Hide, and Show Parts ... 124
Renaming Parts ... 125

Structure Modeling Concepts 126

Import a Hull Model from ShipCAM ... 126

Planar Groups ... 126
Create a Planar Group Drawing Template ... 127
Create Planar Group Drawings ... 127
Create Curved Model Drawings ... 132
Edit Planar Group Properties ... 133
Delete a Planar Group ... 133
Transfer Objects from One Planar Group Drawing to Another ... 133
Transfer Objects from a Unit Drawing to a Planar Group Drawing ... 134
Create a UCS Out of Plane with the Current UCS ... 136
Create a UCS Parallel to the Current UCS ... 137
Activate a UCS ... 138
Select a UCS Viewpoint ... 139
Undo ... 140

Structural Modeling Layers ... 140
Activate a Layer ... 140
Deactivate a Layer ... 141
Copy Objects from One Layer to Another ... 141
Move Objects from One Layer to Another ... 141

Object Display ... 141
Hide or Show Objects ... 141
Display Objects in Wireframe, Hidden Line, or Shaded Mode ... 142
Display Information for an Object within an MLinked Drawing ... 142

Part Views ... 142
Part View Advantages ... 142
Supported ShipConstructor Entities ... 142
Licensing ... 142
Loading Part Views ... 143

Export ... 143
Export a ShipConstructor Drawing to an AutoCAD only Drawing ... 143
Export ShipConstructor Drawings to NavisWorks Drawings ... 144

Model Structure 147

DDROM Best Practices 148
Conceptual Design Stage 148
Maximize DDROM Effectiveness (Relationship Management) 148
Construction Line Management 150
General Use Tips 151

Structural Modeling 151
Database 151
3D Unit Drawings 151
Construction Lines 151
Structure Part Relationships 155
Updating Mirrored and Identical Parts 157
Mark Group Intersections 157

Plate Parts 158
Plate Part Creation 158
Copy a Plate 159
Mirror a Plate 160
Move a Plate 160
Detail a Plate 160

Stiffeners 167
Stiffener Creation 167
Stiffener Edit 169
Stiffener Trimming 170
Stiffener Cutouts 171
Stiffener Attach to Plate Part 174
Stiffener Moldline 174
Stiffener Moving 174

Twisted Stiffeners 175
Twisted Stiffener Edit 176
Twisted Stiffener Trimming 177
Twisted Stiffener Cutouts 177
Twisted Stiffener Moldline 177
Twisted Stiffener Moving 177
Twisted Stiffener Clean Geometry 177

Faceplates 178
Faceplate Creation 178
Faceplate Edit 179
Faceplate Trimming 179
Faceplate Cutouts 179
Faceplate Moldline 179
Faceplate Moving 179

Corrugated Plates 180
Corrugated Plate Creation 180
Corrugated Plate Edit 184

Curved Plates 184
Curved Plate Creation 184
Curved Plate Edit 188
Curved Plate Production Information 189

Planks 189
Planks Creation 190
Planks Edit 191
Adding and Removing Cut Lines to a Plank Collection 192
Splitting Plank Collections 193
Deleting Plank Collections 193

Standard Parts 193

Standard Brackets Attached to Stiffeners 193

Mirror Parts 195
Mirror Parts About the Centerline 195
Mirror Catamaran Hulls 195

Piecemarks 195
Changing a Piecemark Size 195
Change a Piecemark Position 196

Dynamic Marking Blocks 196
Add a Dynamic Marking Block 197
Edit a Dynamic Marking Block 198
Remove a Dynamic Marking Block 199

Part User-Defined Attributes 199

Check Interferences 200
Create an Interference Drawing 200
Checking Interferences 200
Check Local Interferences 203

Replicate Objects to Other Planar Groups 203

Transfer Objects to another Planar Group 204

Miscellaneous Geometry Tasks 205
Insert a Label or Dimension From a Point to a Structural Group 205
Toolpath 205
Dihedral Angle 206
Convert a 3D Object to a 2D Object 207
Project Shapes 207
Reduce the Number of Vertices on Polylines 207
Convert an Ellipse or Spline to a Polyline 208
Set Up Snapping 208

Invalid Parts 209

Check Structure 211

Check Product Hierarchy 211

Check a Unit 211

Update Model and System Drawings 212

Approval Drawings 215

Create an Approval Drawing Template 215
Create an Approval Drawing 216
Insert a Planar Group into an Approval Drawing 216
Save a Bound Approval Drawing 217

Assembly Drawings 218

Set Up Assembly Drawings 219
Set Up an Assembly Drawing Template 219
Set Up an Assembly Drawing BOM Style 223
Select Predefined Templates 223

Generate Assembly Drawings 224
Set Up the Keymap Drawing 224
Generating Assembly Drawings 224

Edit Assembly Drawings 229
Update Parts in an Assembly Drawing 229
Updating an Existing Assembly Drawing 230
Insert Keywords into an Assembly Drawing 231
Update All Keywords 231
BOM Tables 232
Update BOMs 233
Edit BOM Collector Options 233
Viewport Display Options 235
List only Visible 235
Insert a Weld Symbol Table into an Assembly Drawing 235
Update Weld Symbol Tables in an Assembly Drawing 235
Insert an Orientation Icon into an Assembly Drawing 236
Mark an Assembly's CG Position in an Assembly Drawing 236
Change the View in an Assembly Drawing 237
Rotate an Assembly in an Assembly Drawing 237
Insert a Quality Control Matrix into an Assembly Drawing 237
Manual Labeling from BOM 238
Setting up Label Sizes 239

Plot Assembly Drawings 245
Plot an Assembly Drawing 245
Inspect a Plotted Assembly Drawing 246

Labeling 246
Relabel All 246
Manual Label from BOM 246
Copy Label from BOM 246
Relabel from BOM 246
Relabel from Parts 246
Label to part edges 247

BOM Revisions 251
New Revision 253
Delete Revision 253
Delete All Revisions 253

List Revisions....253

Plate Nesting 254

Nesting Setup.... 254
Set Up Nest Options and Colors....254
Set Up a Nest Naming Convention....255
Set Up NC Machines for Nesting....255
Set Up Stocks for Nesting....255
Set Up a Nest BOM Definition....255
Set Up Nest Templates....255

Set Up Nest Drawings and Nests.... 259
Create a Nest Folder....259
Create a Nest Drawing....259
Re-create a Nest Drawing....260
Insert a Nest into a Nest Drawing....261
Insert Parts into a Nest....262
Remove Parts from a Nest....272
Re-nest a Nest....272
Updating Parts in a Nest....273
Assign or Unassign Parts to a Nest....274
Slide a Part Along a Plate Edge or Part Edge....275
Move or Rotate Parts within a Nest....277
Finding Parts in a Nest Drawing....277
Labelling Parts....278
List Un-nested Parts in an Assembly....278
Position Parts Next to each other....279
Hide or Show Nested Parts, Revised Parts, or NC Cutting Paths....280
Insert Keywords into a Nest....281
Edit the Stock Plate of a Nest....281
Delete a Nest....281
Find a Nest....281

Set Up Bridges.... 282
Assign to NC-Machine....284
Insert a Bridge....284
Delete Bridges....285

Set Up Remnants.... 285
Create a Remnant....285
Edit a Remnant....287
Delete a Remnant....288

Check and Fix Nests.... 288
Check Nest Drawings for Errors and Un-nested Parts....288
Update Nest Keywords and BOMs....289
Check Nests for Collisions....289
Re-insert Parts Into a Nest....290

Generate NC Code for Nests.... 290
Export Nests to NC-Pyros....290
Generate NC Code for Nests....292

Export Parts to Individual Drawings....292
Import Cutting Paths Into Nests....292
Update Nest BOMs....293
Plot Nests....294
Generate Nest Reports....294
Prepare Work Package....294
Profile Plots 295
Set Up Profile Plot Templates....296
Profile Plot Drawing Template....296
Profile Sheet Template....296
Profile Plot Template....297
Create a Profile Plot Drawing....299
Set Up Profile Plots....300
Insert Profile Plots Into a Profile Plot Drawing....301
Updating Profile Plots....303
Zoom Into a Profile Plot within a Profile Plot Drawing....304
Print Profile Plots....304
Check Profile Plots....304
Update Profile Plot BOMs....305
Profile Nesting 305
Set Up Profile Stocks....305
Select an Assembly and Stock for Profile Nesting....305
Create or Delete Profile Nests or Remnants....307
Create Empty Profile Nests or Remnants....307
Delete a Profile Nest or Remnant....308
Add and Remove Parts to and from Profile Nests....309
Manually Add Parts to Profile Nests....309
Automatically Create Profile Nests for Selected Parts....310
Remove Parts From Profile Nests....311
Changing Parts and Stocks and Checking Profile Nests....311
Issue and Un-issue Profile Nests....312
Issue Profile Nests....312
Un-issue Profile Nests....313
Generate Profile Nest Reports....313
Generate a Summary Report or Detailed Report....314

Generate a Stock Usage Report....315

Set Profile Nests to Cut or Un-cut....315
Set Profile Nests to Cut....315
Set Profile Nests to Un-cut....316

Part Property Labels 316
Placing Field Labels....318
Copying Field Labels....318

Leader Distribution Lines 319
Set Leader Insertion Mode....321
Redistribute Leaders....321

Export To Dwg 321

Exporting using the Graphic User Interface....322

Command Line Export....324

Fix Duplicate Drawing Names 325

Command line Fix Duplicate....325

Update Production Drawings 325

Appendix A: Menus, Tools & Commands 327

Manager Menus....327

Structure Menu....342
Structure > Profiles > Endcut Catalog....358

Toolbars....363
Equipment Toolbar....363
Hull Toolbar....363
Hull Curve Toolbar....363
Hull Expand Toolbar....363
Hull Pin Jig Toolbar....363
Hull Porcupine Toolbar....363
Hull Stringer Toolbar....364
Hull Surface Toolbar....364
Labels Toolbar....364
Penetrations Toolbar....364
ShipConstructor Toolbar....364
Structural Tools Toolbar....364
Structure Toolbar....364
Structure Plate Flyout....365
Structure Flange Plate Flyout....365

Structure Green Flyout 365
Structure Part Objects Flyout 365
Structure Corner Treatment Flyout 366
Structure Stiffener Flyout 366
Structure Faceplate Flyout 366
Structure Display Toolbar 367
Utilities Toolbar 367
Layers Flyout 367

ShipConstructor Menu 368
ShipConstructor > Navigator 368
ShipConstructor > Manager 383
ShipConstructor > Project > New Project 383
ShipConstructor > Project > Copy Project 383
ShipConstructor > Model Link 383
ShipConstructor > Planar Group > Transfer Objects to Group 384
ShipConstructor > Penetration Manager 384
ShipConstructor > Check > Check Project 384
ShipConstructor > Check > Check Unit 384
ShipConstructor > Check > Check Templates 385
ShipConstructor > Check > Fix Duplicate Drawing Names 385
ShipConstructor > Update Model and System Drawings 385
ShipConstructor > Update Production Drawings 386
ShipConstructor > Product Hierarchy 386
ShipConstructor > Profile Nesting 386
ShipConstructor > Revisions 388
ShipConstructor > Licensing 390
ShipConstructor > ShipConstructor Help 392
ShipConstructor > About ShipConstructor 392

SC Structure Menu 393
SC Structure > Mark Group Intersections 393
SC Structure > Show / Hide Options 394
SC Structure > Plate > New 397
SC Structure > Plate > Edit 400
SC Structure > Plate > Add Flange 401
SC Structure > Plate > Edit Flange 402
SC Structure > Plate > Add Green 403
SC Structure > Plate > Edit Green 403
SC Structure > Plate > Add Objects 404
SC Structure > Plate > Remove Objects 406
SC Structure > Plate > Manage Cutouts 406
SC Structure > Plate > Corner Treatment > Insert 407
SC Structure > Plate > Corner Treatment > Remove 408
SC Structure > Plate > Corner Treatment > Display 408
SC Structure > Plate > Bevel > Create 409
SC Structure > Plate > Bevel > Edit 409
SC Structure > Plate > Bevel > Remove 409
SC Structure > Plate > Manage Datum Lines 410
SC Structure > Plate > Manage Weld Shrinkage Icon 410
SC Structure > Plate > Invalid Boundary Diagnostic 411
SC Structure > Stiffener > New 412
SC Structure > Stiffener > New from Points 414

SC Structure > Stiffener > Edit 414
SC Structure > Stiffener > Attach to Plate 414
SC Structure > Stiffener > New Twisted 415
SC Structure > Stiffener > Edit Twisted 416
SC Structure > Stiffener > Clean Geometry of Twisted 416
SC Structure > Stiffener > Cutout > Insert 416
SC Structure > Stiffener > Cutout > Edit 417
SC Structure > Stiffener > Trim 418
SC Structure > Stiffener > Insert Weld Seam Reliefs 419
SC Structure > Stiffener > Extract Neutral Axis 419
SC Structure > Stiffener > Extract Mold Line 420
SC Structure > Faceplate > New 420
SC Structure > Faceplate > Edit 421
SC Structure > Faceplate > Copy and Rotate 180 422
SC Structure > Faceplate > Cutout > Insert 422
SC Structure > Faceplate > Cutout > Edit 422
SC Structure > Faceplate > Trim 422
SC Structure > Faceplate > Extract Neutral Axis 423
SC Structure > Faceplate > Extract Mold Line 423
SC Structure > Corrugated Plate > New 423
SC Structure > Corrugated Plate > Edit 425
SC Structure > Corrugated Plate > Extract Cross-section 425
SC Structure > Curved Plate > New 426
SC Structure > Curved Plate > Edit 427
SC Structure > Curved Plate > Define New Outer Toolpath 428
SC Structure > Curved Plate > Extract Production Info 428
SC Structure > Curved Plate > Import Curved Plate From Rhino 428
SC Structure > Planks > New 428
SC Structure > Planks > Edit Collection 430
SC Structure > Planks > Split Plank Collection 430
SC Structure > Planks > Delete Plank Collection 430
SC Structure > Parts > Edit 430
SC Structure > Parts > List 431
SC Structure > Parts > Find 432
SC Structure > Parts > Relationships 432
SC Structure > Parts > Information 433
SC Structure > Parts > Edit Component 435
SC Structure > Parts > Extract Components 438
SC Structure > Parts > Add Manual Cutouts 438
SC Structure > Parts > Add Dynamic Marking Block 438
SC Structure > Parts > Boundary Diagnostic 439
SC Structure > Construction Line > Information 439
SC Structure > Construction Line > Relationships 440
SC Structure > Construction Line > Convert to User Construction Line 441
SC Structure > Construction Line > Edit Offset 441
SC Structure > Construction Line > Clear Trims 441
SC Structure > Construction Line > Swap Construction Line 441
SC Structure > Construction Line > Replace Construction Line Geometry 442
SC Structure > Construction Line > Add Bevel Angles 442
SC Structure > Construction Line > Show Bevel Angles 443
SC Structure > Construction Line > Hide Bevel Angles 444
SC Structure > Construction Line > Clear Bevel Angles 444
SC Structure > Insert Standard Part 444

SC Structure > Convert To Structure Part 444
SC Structure > Insert Standard Part at Stiffener 445
SC Structure > Transfer Objects To Group 449
SC Structure > Move Planar Group 449
SC Structure > Replicate Objects to Other Group 449
SC Structure > Check Group DWG 450
SC Structure > Show Unused Objects 451

SC Utilities Menu 451
SC Utilities > 3D Viewpoint 451
SC Utilities > Activate UCS 451
SC Utilities > UCS > Flip UCS X 453
SC Utilities > UCS > Flip UCS Y 453
SC Utilities > UCS > Swap UCS XY Axis 454
SC Utilities > Hide Objects 454
SC Utilities > Unhide Objects 454
SC Utilities > Unhide All Objects 454
SC Utilities > Clip Current View 455
SC Utilities > Remove Clip 456
SC Utilities > 3D to 2D 457
SC Utilities > Orthographic Projection 458
SC Utilities > Remove Vertices Below Tolerance 458
SC Utilities > Convert Ellipse/Spline to Polyline 459
SC Utilities > Layer > Activate 459
SC Utilities > Layer > Deactivate 459
SC Utilities > Layer > Copy Geometry to 460
SC Utilities > Layer > Move Geometry to 460
SC Utilities > Toolpath 460
SC Utilities > Fillet 461
SC Utilities > Mirror about Centerline 461
SC Utilities > Dihedral Angle 461
SC Utilities > Reload Drawing 462
SC Utilities > Create Quality Matrix 462
SC Utilities > Property Labels > Property Label 465
SC Utilities > Property Labels > Copy Field Label 465
SC Utilities > Property Labels > Copy Field Label Quick 465
SC Utilities > Property Labels > Edit Field Label 465
SC Utilities > Property Labels > Replace Object References 465
SC Utilities > Property Labels > Set Single Click Field Label 466
SC Utilities > List Item within Block/Xref 466
SC Utilities > Check Local Interferences 466
SC Utilities > Export > Export to DWG 466
SC Utilities > Export > Export to NWD 467
SC Utilities > Export > Export to NWC 467
SC Utilities > Snap 467
SC Utilities > Random Color 467
SC Utilities > ShipCAM > Import Files 468
SC Utilities > ShipCAM > Convert IGES to Mesh 468

Command Line Commands 468
Plate Part Extract Polyline 468
Create Boundary Polyline 468
Create Boundary Polyline Debug 469

Create Boundary Polyline from Outside Pickpoint 469
Show Plate Part Construction Lines 469
Hide Plate Part Construction Lines 469
Show Plate Part Construction Lines Only 469
Show Plate Part and Its Construction Lines Only 470
Show Extended Data 470
Fix Identical Part Names 470
Clean Geometry of all Twisted Stiffeners 470
Set Dynamic Marking Block Text Static 471
Set Dynamic Marking Block Text Normal 471
Copy Product Hierarchy 471
Export Product Hierarchy 471
Import Product Hierarchy 471
Part View Load via Extents – SCPartViewLoadByExtents 472
Part View Load via Selected Object Extents – SCPartViewLoadBySelectedExtents 472
Part View Load Associated Structure Parts – SCPartViewLoadAssociated 472
Part View Delete All Part Views in Drawing – SCPartViewDeleteAll 472

Diagnostic Commands 473
Command Trace - Enable 473
Command Trace - Disable 473
Debug Trace - Enable 473
Debug Trace - Log 473
Debug Trace - Disable 474
Performance Monitor - Start 474
Performance Monitor - End 474
Performance Monitor - Display 474
Save Structure Drawing State 474
Load Structure Drawing State from File 475
DataManager Dump 475

Appendix B: Structure Production Command Reference 477

Toolbars 477
Assembly Toolbar 477
Leader Utils Toolbar (page **Error! Bookmark not defined.**) 478
Nest Toolbar 478

Ribbon Tabs 479
Nest and Profile Plots Tab 479
Structure Production Tab 482
Distributed Systems Production Tab 485

SC Approval Menu 488
SC Approval > Insert Group 488
SC Approval > Save As Bound Approval 489

SC Assembly Menu 489
SC Assembly > Update Drawing 490
SC Assembly > Orientation Icon 490
SC Assembly > CG Point 491
SC Assembly > Insert Keywords 491
SC Assembly > Update All Keywords 492

SC Assembly > Structure Drawing Options....492
SC Assembly > Pipe Drawing Options....492
SC Assembly > HVAC Drawing Options....492
SC Assembly > Equipment Drawing Options....493
SC Assembly > Viewport Options....493
SC Assembly > Set Viewport Display Options....493
SC Assembly > Save Drawing Options....494
SC Assembly > Load Drawing Options....494
SC Assembly > Save Viewport Display Options....494
SC Assembly > Load Viewport Display Options....494
SC Assembly > Insert BOM Table....495
SC Assembly > Update BOMs....495
SC Assembly > Edit Collector Options....495
SC Assembly > Toggle List Only Visible....495
SC Assembly > Label > Manual Label from BOM....496
SC Assembly > Label > Copy Label from BOM....496
SC Assembly > Label > Relabel All....496
SC Assembly > Label > Relabel from BOM....497
SC Assembly > Label > Relabel from Parts....497
SC Assembly > Label > Label on Visible Edge On/Off....497
SC Assembly > Label > Curved Plates in Visible Edge Detection On/Off....497
SC Assembly > Label > Adjacent Part Edge Tolerance....498
SC Assembly > Label > Edge Length Filter....498
SC Assembly > Label > Edge Determination Minimum Angle....498
SC Assembly > Label > Label Reset Automatic Settings....498
SC Assembly > BOM Revisions > New BOM Revision in Current Layout....499
SC Assembly > BOM Revisions > New BOM Revision in All Layouts....499
SC Assembly > BOM Revisions > Delete BOM Revision in Current Layout....499
SC Assembly > BOM Revisions > Delete BOM Revision in All Layouts....499
SC Assembly > BOM Revisions > Delete All BOM Revisions in Current Layout....500
SC Assembly > BOM Revisions > Delete All BOM Revisions in All Layouts....500
SC Assembly > BOM Revisions > List BOM Revisions....500
SC Assembly > Adjust Leader Spacing....500
SC Assembly > Attach to Viewport Tracking....501
SC Assembly > Create Leader Distribution Line....501
SC Assembly > Set Leader Insertion Mode....501
SC Assembly > Redistribute Leaders....502
SC Assembly > Transfer Leaders to Other Line....502
SC Assembly > Distribution line merge distance tolerance....502
SC Assembly > Distribution line merge angle tolerance....502
SC Assembly > Insert Weld Symbol Table....503
SC Assembly > Update Weld Symbol Tables....503
SC Assembly > Global Dimension to Point....503

SC Assembly Template Menu....503
SC Assembly Template > Orientation Icon....504
SC Assembly Template > Insert Keywords....505
SC Assembly Template > Structure Drawing Options....505
SC Assembly Template > Pipe Drawing Options....506
SC Assembly Template > HVAC Drawing Options....506
SC Assembly Template > Equipment Drawing Options....506
SC Assembly Template > Viewport Options....506
SC Assembly Template > Set Viewport Display Options....507

SC Assembly Template > Insert Weld Symbol Table 507
SC Assembly Template > Insert BOM Table 507
SC Assembly Template > Edit Collector Options 507
SC Assembly Template > Toggle List Only Visible 508
SC Assembly Template > Update from a Previous Version 508

SC Product Hierarchy Menu 508
SC Product Hierarchy > Develop Product Hierarchy 508
SC Product Hierarchy > Update Drawing 509

SC Interference Menu 509
SC Interference > Check Interferences 510
SC Interference > Update Drawing 510

SC Plate Nest Menu 510
SC Plate Nest > Nest > New 511
SC Plate Nest > Nest > Delete 512
SC Plate Nest > Nest > Find 512
SC Plate Nest > Nest > Re-Nest 512
SC Plate Nest > Nest > Edit Stock Plate 513
SC Plate Nest > Nest > Plot 513
SC Plate Nest > Nest > Assign to NC-Machine 513
SC Plate Nest > Remnant > New 513
SC Plate Nest > Remnant > Delete 514
SC Plate Nest > Remnant > Edit Remnant 514
SC Plate Nest > Remnant > Edit Remnant Label 514
SC Plate Nest > Part > Insert 514
SC Plate Nest > Part > Re-insert 515
SC Plate Nest > Part > Remove 515
SC Plate Nest > Part > Slide Along 515
SC Plate Nest > Part > Move/Rotate 515
SC Plate Nest > Part > Align 516
SC Plate Nest > Part > Rotate Shortest Dimension 516
SC Plate Nest > Part > List 517
SC Plate Nest > Part > Find 517
SC Plate Nest > Part > Snap 517
SC Plate Nest > Part > Align with H or V 517
SC Plate Nest > Part > List Un-nested Parts 518
SC Plate Nest > Part > Export Nest Parts to DWG 518
SC Plate Nest > Part > Export Nest Parts to DXF 518
SC Plate Nest > Layers > Visibility 518
SC Plate Nest > Layers > Activate All 519
SC Plate Nest > Layers > Flip NC-Pyros 519
SC Plate Nest > Insert Keywords 519
SC Plate Nest > Auto Label from BOM Index 519
SC Plate Nest > Label Properties 520
SC Plate Nest > Nest Manager 520
SC Plate Nest > Check Nests and BOM 521
SC Plate Nest > Update Nests and BOM 522
SC Plate Nest > Nest Collision Check 522
SC Plate Nest > NC-Pyros > Export To 523
SC Plate Nest > NC-Pyros > Import From 525
SC Plate Nest > Bridge > New 525
SC Plate Nest > Bridge > Delete 525

SC Plate Nest > Bridge > Update 526

SC Plate Nest Template Menu 526
SC Plate Nest Template > Define Border 526
SC Plate Nest Template > Insert Keywords 526
SC Nest Template > Insert BOM Table 529
SC Plate Nest Template > Update from a Previous Version 529
SC Plate Nest Template > Check Drawing 529

SC Profile Plots Menu 529
SC Profile Plots > Insert Profile Plots 529
SC Profile Plots > Options 532
SC Profile Plots > Zoom 536
SC Profile Plots > Print Plots 537
SC Profile Plots > Check Profile Drawings 538
SC Profile Plots > Update All BOMs 539

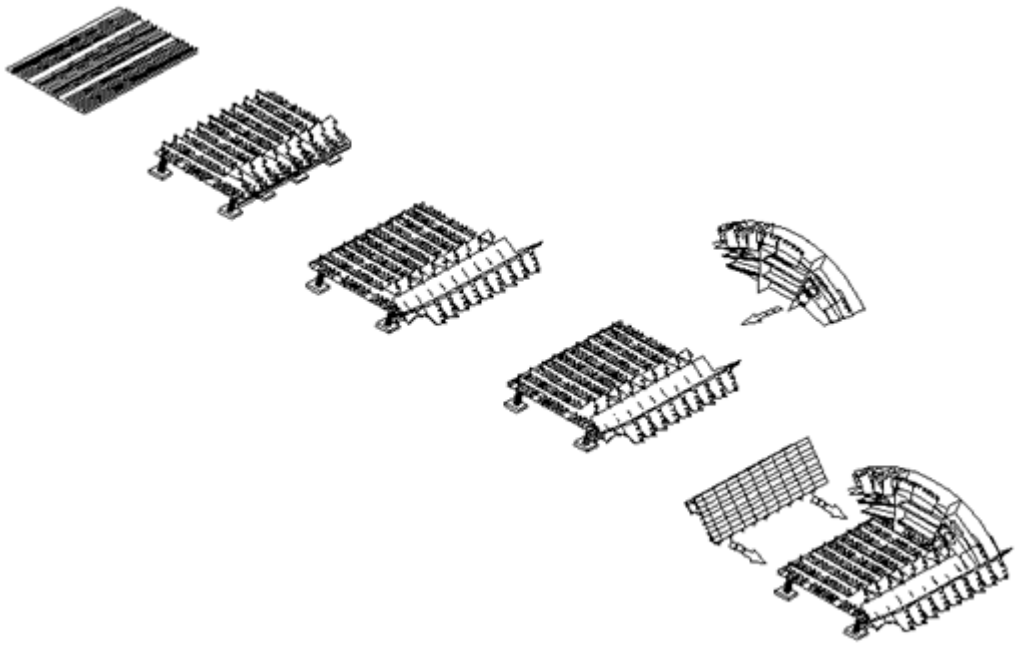
SC Profile Plot Template Menu 539
SC Profile Plot Template > Draw Border 539
SC Profile Plot Template > Draw Web View Area 540
SC Profile Plot Template > Insert Keywords 540
SC Profile Plot Template > Insert BOM Table 542
SC Profile Plot Template > Replace Legacy Keywords 543

SC Profile Sheet Template Menu 543
SC Profile Sheet Template > Draw Border 543
SC Profile Sheet Template > Draw Area 543
SC Profile Sheet Template > Insert Keywords 544
SC Profile Sheet Template > Replace Legacy Keywords 544

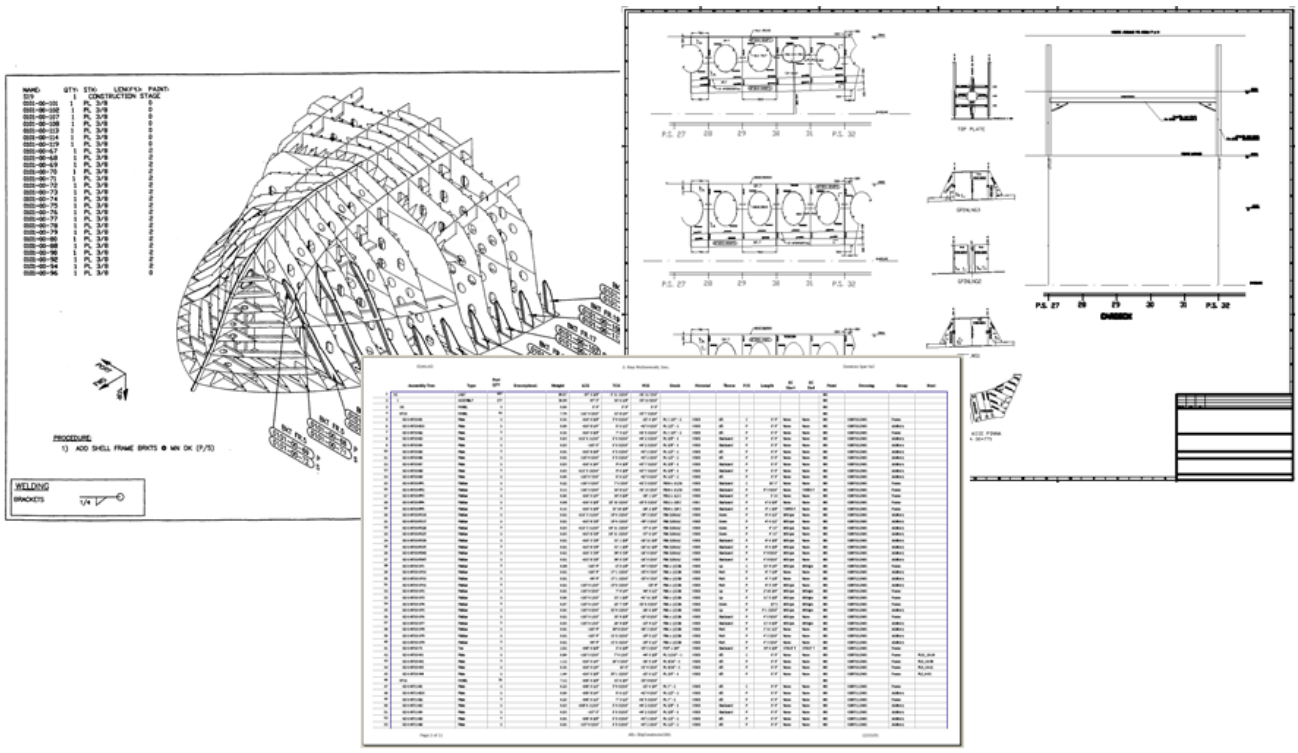
Index 545

ShipConstructor

ShipConstructor is a set of software tools for planning the production of ships and offshore structures, designing and modeling hull, structure, piping, HVAC, penetrations, and equipment, and generating production documentation to enable efficient fabrication.



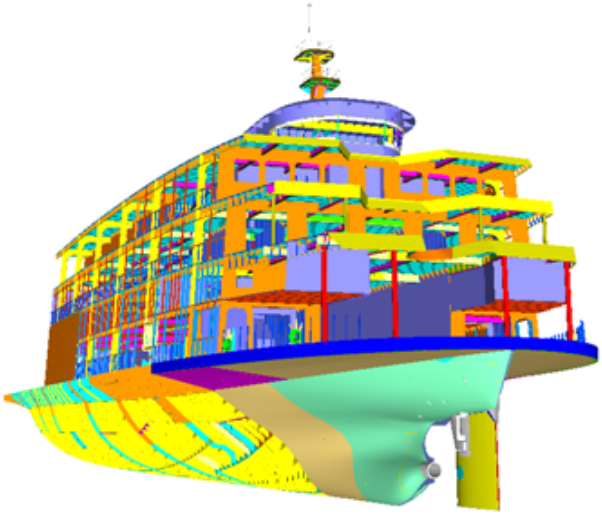
Assembly sequence



Production output

Note: ShipConstructor consists of several modules that you can purchase separately. This document describes features in various ShipConstructor modules. If you do not have a license for all ShipConstructor modules, you may not be able to use some of the features described in this document. For more information on ShipConstructor modules and licensing, visit www.ShipConstructor.com.

Projects, Models, Drawings and Libraries
All work in ShipConstructor is done on a specific project. A project represents a single vessel. As you work on a project, you build a three-dimensional digital model of the vessel.



3D product model courtesy of Marinette Marine Corp. and Genoa Design International

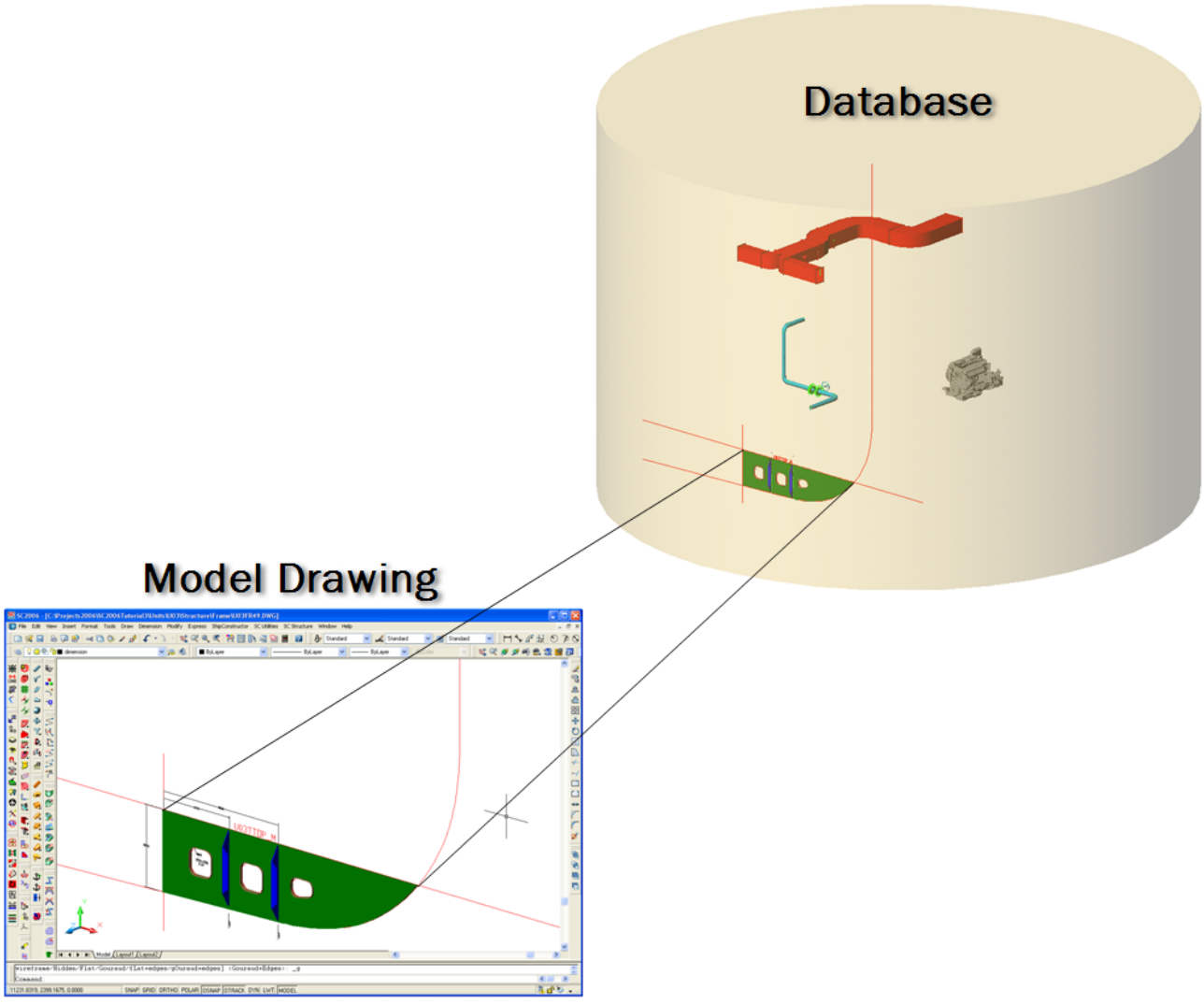
Database Driven Relational Object Model (DDROM)

ShipConstructor uses a fundamentally new technology for creating, storing, and updating part information: the Database Driven Relational Object Model (DDROM).

DDROM is similar to parametric modeling. Parametric modeling is a way of defining geometry in which geometric dependencies are built-in, so that editing one shape will cause other shapes to change size or location.

For example, when you update a part after it has been defined, any objects related to the part that are affected by the update are also modified.

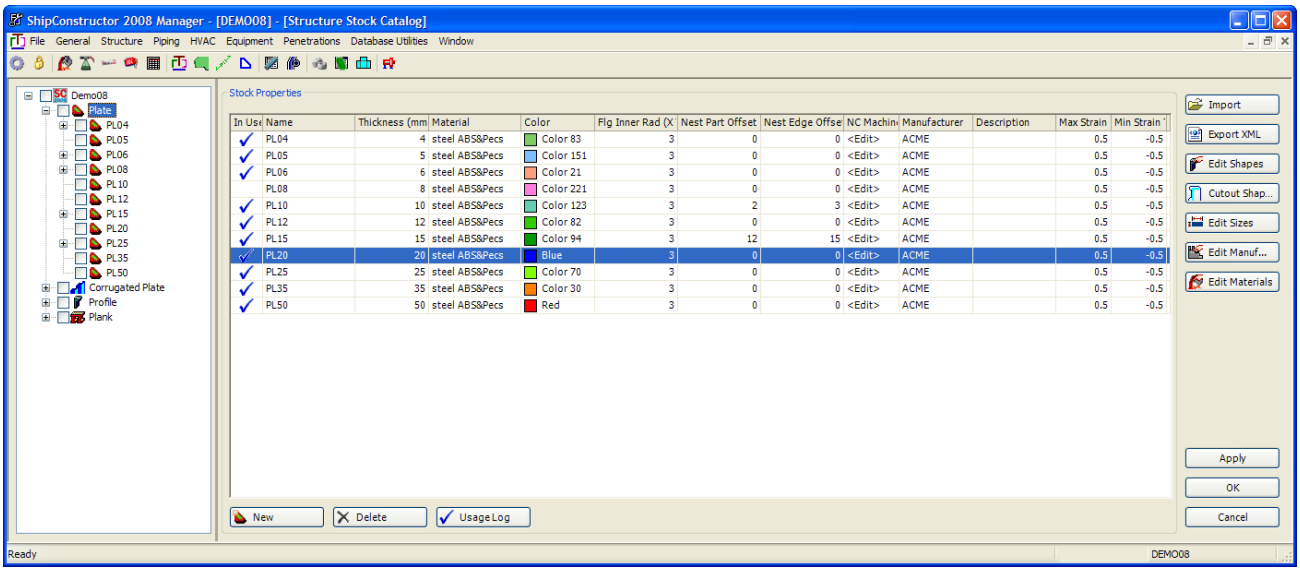
When you work in ShipConstructor, you create and work within drawings. Each drawing is only an interface between you and the model that is stored in the database. The model is a three-dimensional representation of the entire vessel. Each project contains many model drawings, but only one model.



Objects in the database

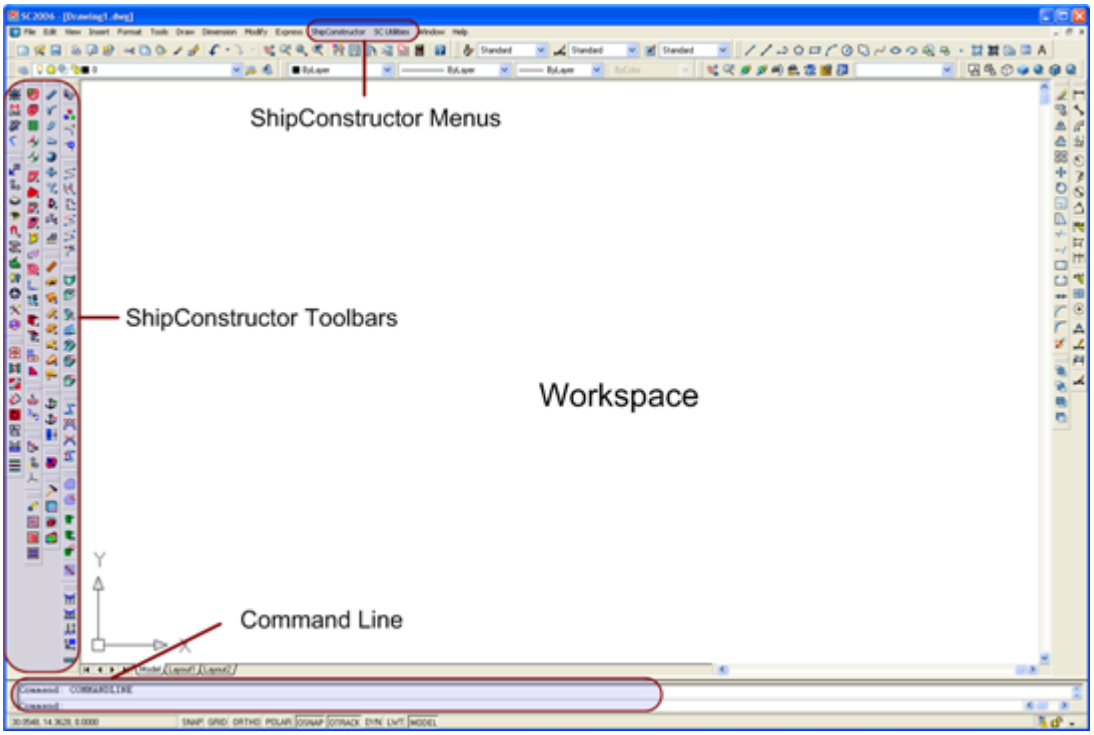
Manager

Another part of each project is the Manager. Manager is where you set up stock items and other rules or conventions.

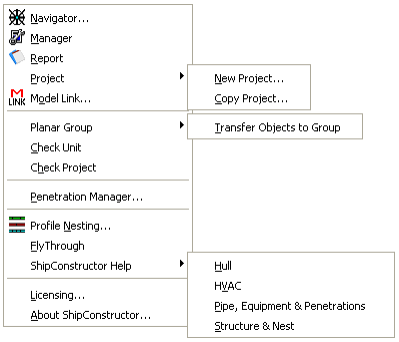

ShipConstructor 2008 Manager - [DEMO08] - [Structure Stock Catalog]
File General Structure Piping HVAC Equipment Penetrations Database Utilities Window
Demo08
Plate
PL04
PL05
PL06
PL08
PL10
PL12
PL15
PL20
PL25
PL35
PL50
Corrugated Plate
Profile
Plank
Stock Properties
In Use Name Thickness (mm Material Color Flg Inner Rad (X Nest Part Offset Nest Edge Offse NC Machin Manufacturer Description Max Strain Min Strain
PL04 4 steel ABS&Pecs Color 83 3 0 0 <Edit> ACME 0.5 -0.5
PL05 5 steel ABS&Pecs Color 151 3 0 0 <Edit> ACME 0.5 -0.5
PL06 6 steel ABS&Pecs Color 21 3 0 0 <Edit> ACME 0.5 -0.5
PL08 8 steel ABS&Pecs Color 221 3 0 0 <Edit> ACME 0.5 -0.5
PL10 10 steel ABS&Pecs Color 123 3 2 3 <Edit> ACME 0.5 -0.5
PL12 12 steel ABS&Pecs Color 82 3 0 0 <Edit> ACME 0.5 -0.5
PL15 15 steel ABS&Pecs Color 94 3 12 15 <Edit> ACME 0.5 -0.5
PL20 20 steel ABS&Pecs Blue 3 0 0 <Edit> ACME 0.5 -0.5
PL25 25 steel ABS&Pecs Color 70 3 0 0 <Edit> ACME 0.5 -0.5
PL35 35 steel ABS&Pecs Color 30 3 0 0 <Edit> ACME 0.5 -0.5
PL50 50 steel ABS&Pecs Red 3 0 0 <Edit> ACME 0.5 -0.5
Import
Export XML
Edit Shapes
Cutout Shap...
Edit Sizes
Edit Manuf...
Edit Materials
Apply
OK
Cancel
New
Delete
Usage Log
Ready
DEMO08

Workspace and Tools

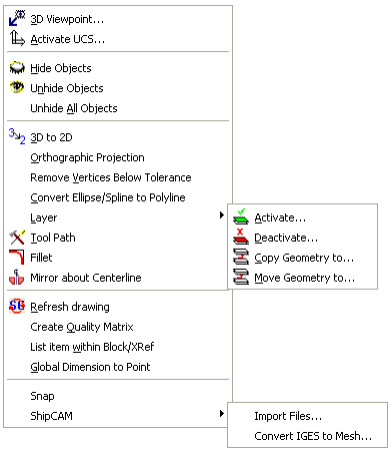
Most ShipConstructor tools are available within AutoCAD from special menus and toolbars or by typing commands in the command line.



The ShipConstructor menu and SC Utilities menu are always available. Other menus are also available depending on the type of drawing you have open.



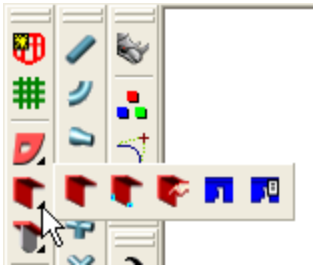
ShipConstructor menu



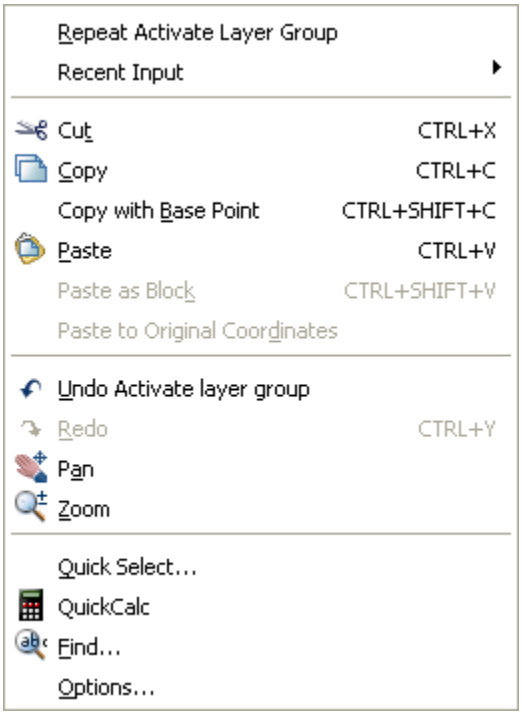
SC Utilities menu

The toolbars available to you when using ShipConstructor are controlled by the current AutoCAD Workspace you have loaded. ShipConstructor provides several predefined workspaces; ShipConstructor, Pipe, HVAC, and Structure. Loading a given workspace can be done using the AutoCAD **Workspaces** toolbar, the **workspace** command entry, or the **wssettings** command entry. More information on using workspaces can be found in the AutoCAD help documentation.

Toolbar buttons with a small black triangle let you access a Flyout toolbar containing more tools.



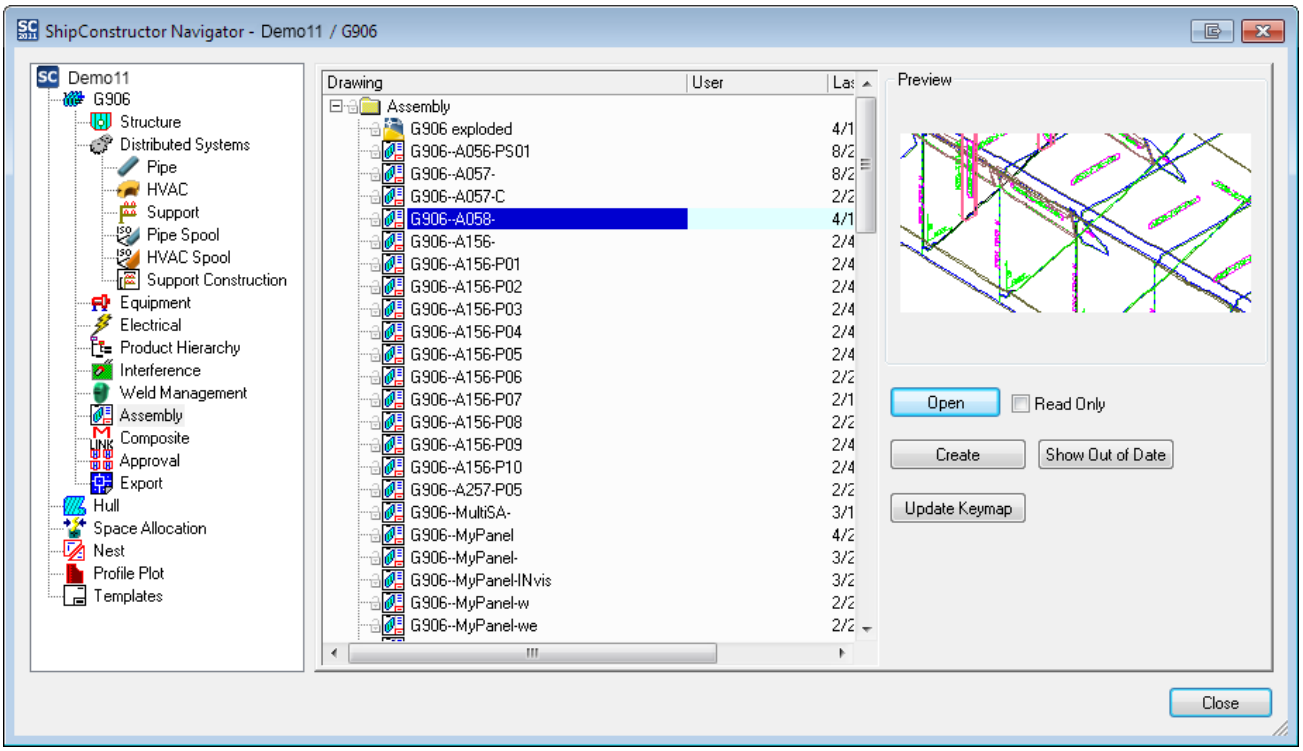
You can also access certain ShipConstructor tools by clicking the right mouse button. The menu that appears depends on the type of object you select or click on. These menus are called right-click menus (or sometimes shortcut menus).



You can also use many of AutoCAD's tools when working on a ShipConstructor project.

Some ShipConstructor tools run independently of AutoCAD.

Navigator
One of the most frequently used tools is Navigator. Navigator lets you open and create drawings and is found at ShipConstructor > Navigator (page 368).



Getting Started

This section describes how to start running ShipConstructor and how to perform basic administrative tasks in ShipConstructor.

Start ShipConstructor

To start ShipConstructor

1. Do either of the following:
   - From the Windows Start button, choose Start > All Programs > ShipConstructor2012 > ShipConstructor2012.
   - On the Windows Desktop, double-click the ShipConstructor shortcut icon. Do not use the AutoCAD shortcut.

   Make sure you see the ShipConstructor and SC Utilities menu in the menu bar. If not, contact your system administrator, or see the Installation Guide for troubleshooting information.

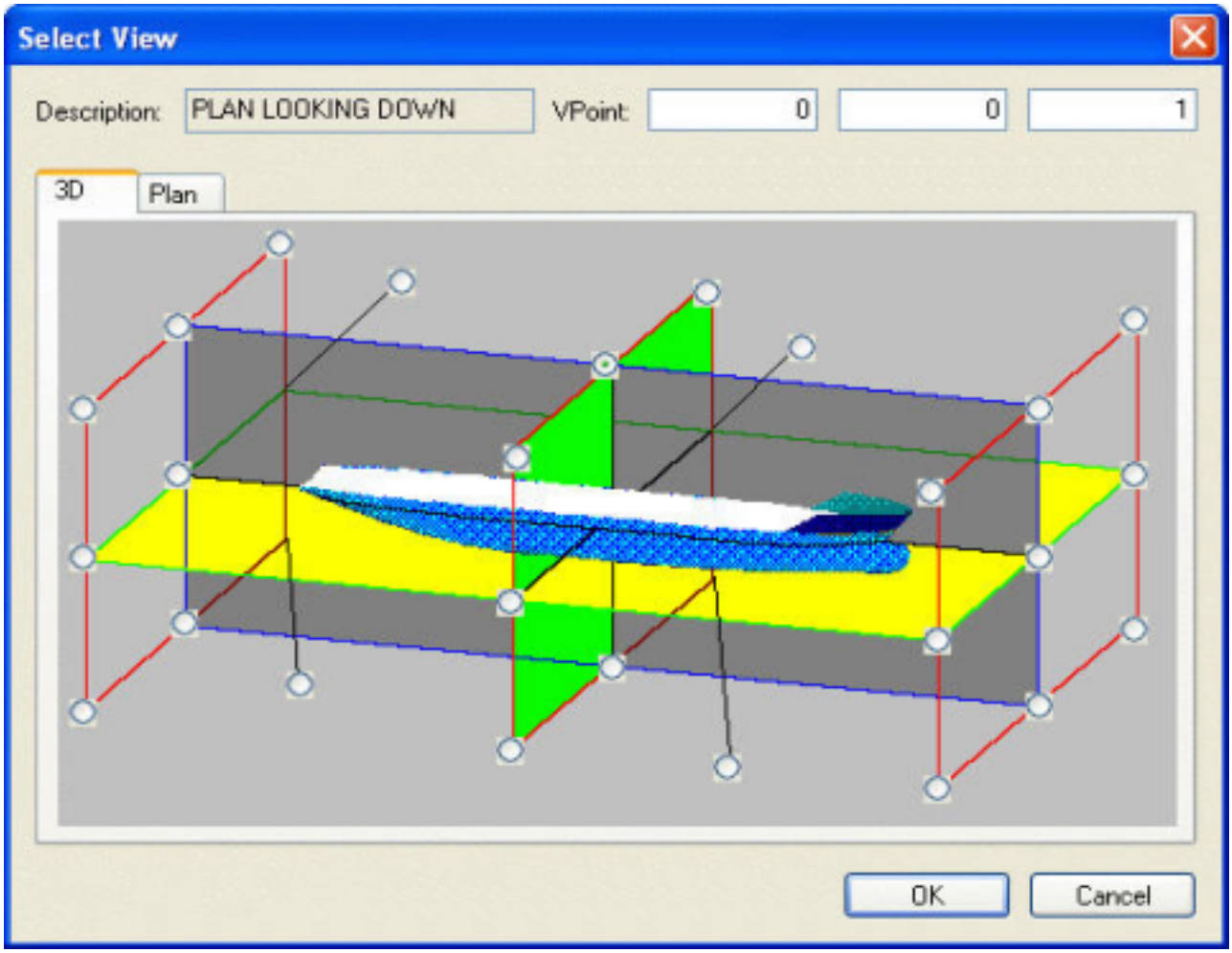
Check Your System Font

ShipConstructor's windows are designed for the small system font (96 dpi). If you are using the large system font, some ShipConstructor windows will not display properly.

To check your system font

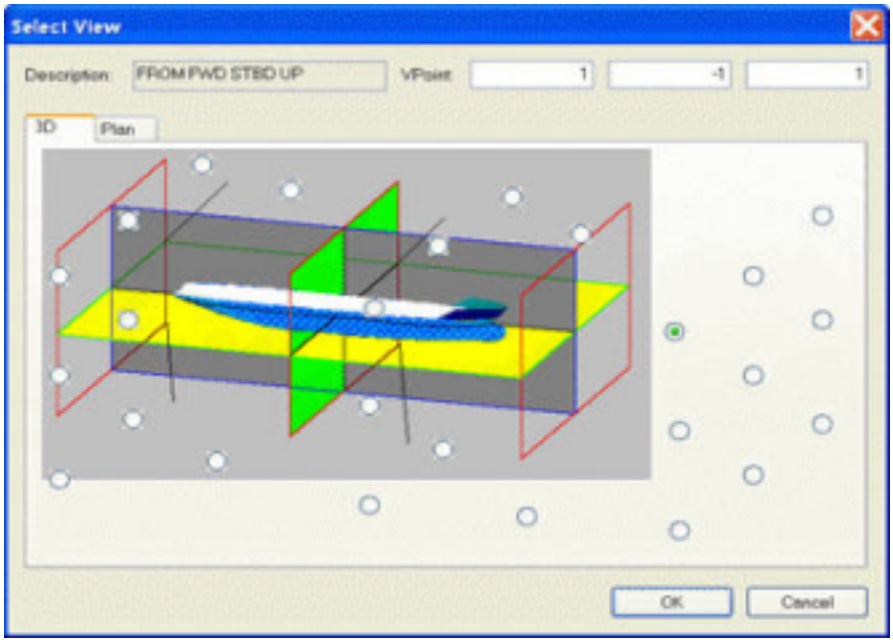
1. Choose SC Utilities > 3D Viewpoint (page 451) to open the Select View window.

   If you are using the small system font, the circles are aligned with the 3D grid points.



Small system font

If you are using the large system font, the circles are misaligned with the grid.

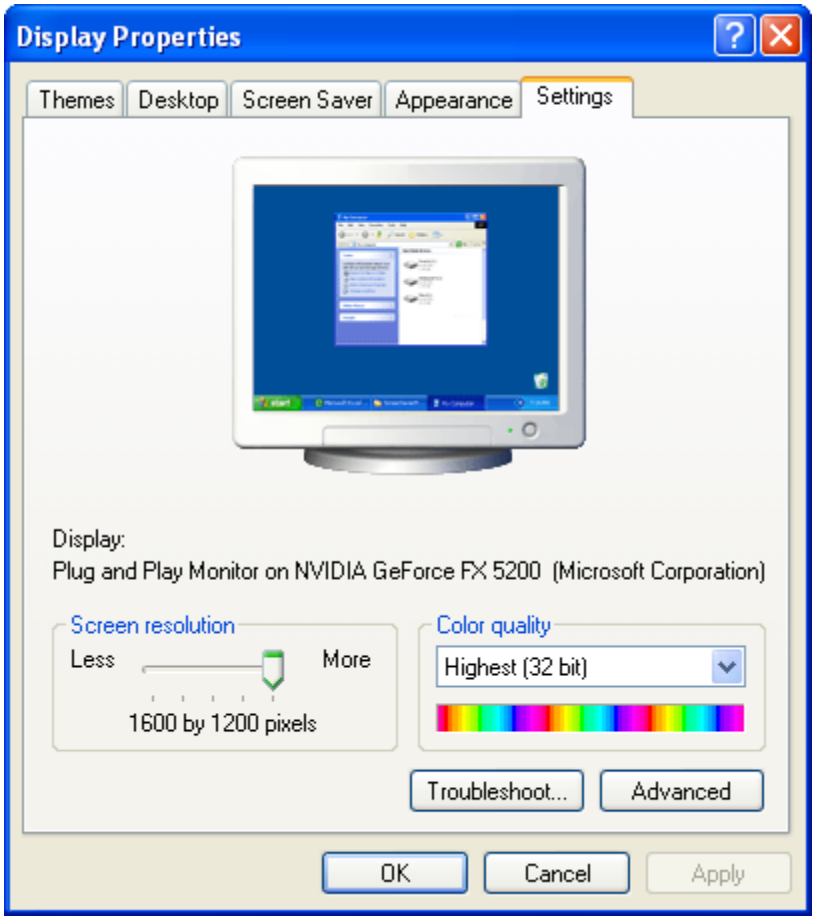


Large system font

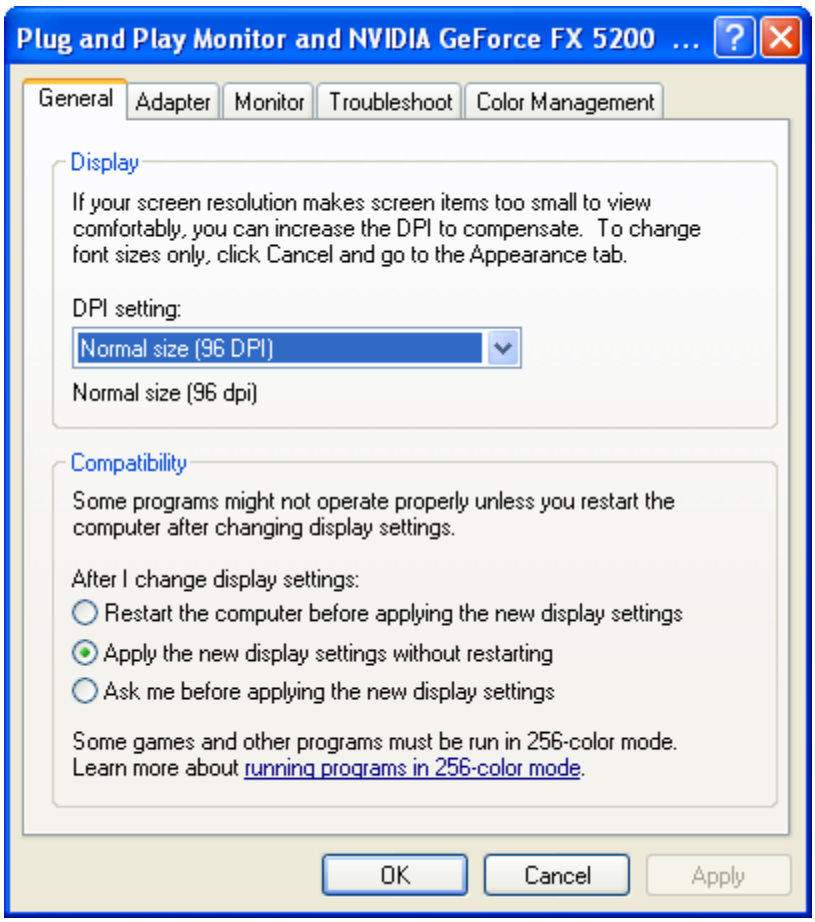
To set your system font

Note: The following instructions are for Windows XP and may be slightly different for other operating systems.

1. Right-click on your Windows Desktop and choose Properties (or from the Windows Start menu, choose Start > Control Panel to open the Control Panel, then double-click the Display icon) to open the Display Properties window.
2. Select the Settings tab.



3. Click Advanced to open the options window for your display device.


Plug and Play Monitor and NVIDIA GeForce FX 5200 ...
General
Adapter
Monitor
Troubleshoot
Color Management
Display
If your screen resolution makes screen items too small to view comfortably, you can increase the DPI to compensate. To change font sizes only, click Cancel and go to the Appearance tab.
DPI setting:
Normal size (96 DPI)
Normal size (96 dpi)
Compatibility
Some programs might not operate properly unless you restart the computer after changing display settings.
After I change display settings:
Restart the computer before applying the new display settings
Apply the new display settings without restarting
Ask me before applying the new display settings
Some games and other programs must be run in 256-color mode.
Learn more about running programs in 256-color mode.
OK
Cancel
Apply

4. Set DPI Setting to Normal Size (96 DPI).
5. Click OK.

   Note: You may need to re-start your computer. You may also need your Windows installation CD if small fonts are not already loaded on your hard drive.

   Check your system font (see above) again to make sure ShipConstructor windows display properly.

   Note: If text is too small for you to read, reduce the resolution of your display. In the Display Properties window, select the Settings tab and set Screen resolution to a lower value. For example, if your current Screen resolution is 1280 x 1024, reduce it to 1024 x 768. However, do not use a Screen resolution less than 1024 x 768.

Open a Project

Note: For more information on project setup and administration, see the Project Management manual.

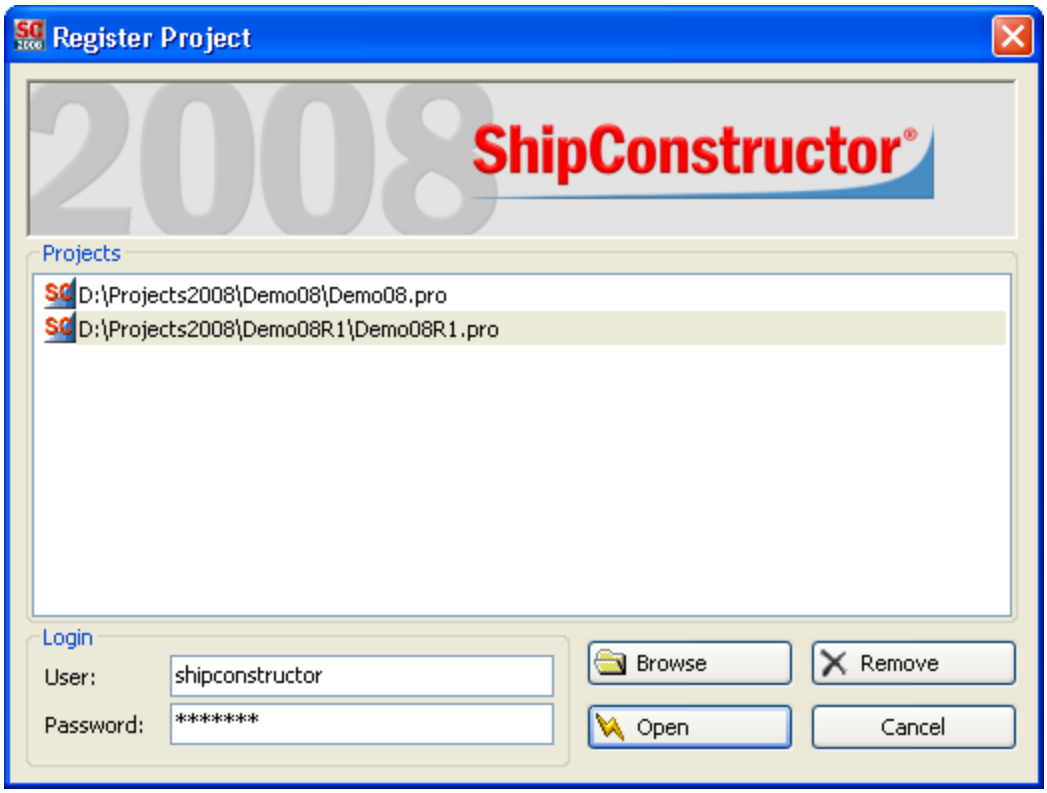
To open a project

Note: You can also open a project from Manager by choosing File > Change Project.

1. Choose ShipConstructor > Navigator to open Navigator.

   If you do not currently have a project open (for example, after initially starting ShipConstructor), the Register Project window appears.

   If you currently have a project open, Navigator appears. Select the project from the component list (for example, SC2009Demo) and click Change Project to open the Register Project window.



2. Select the project from the list or click Browse to locate a project file. (A ShipConstructor project file has a *.pro extension.)
3. Enter your User name and Password.

   Note: For the project templates, the User name is ShipConstructor and the Password is shipcon.

   If you do not have a user account, ask your system administrator to set one up, or see User Accounts in the Project Management manual for details.
4. Click Open.

   Note: If this is the first time you open a project after starting ShipConstructor, the Licensing window appears. See Get Licenses (page 12) for details.

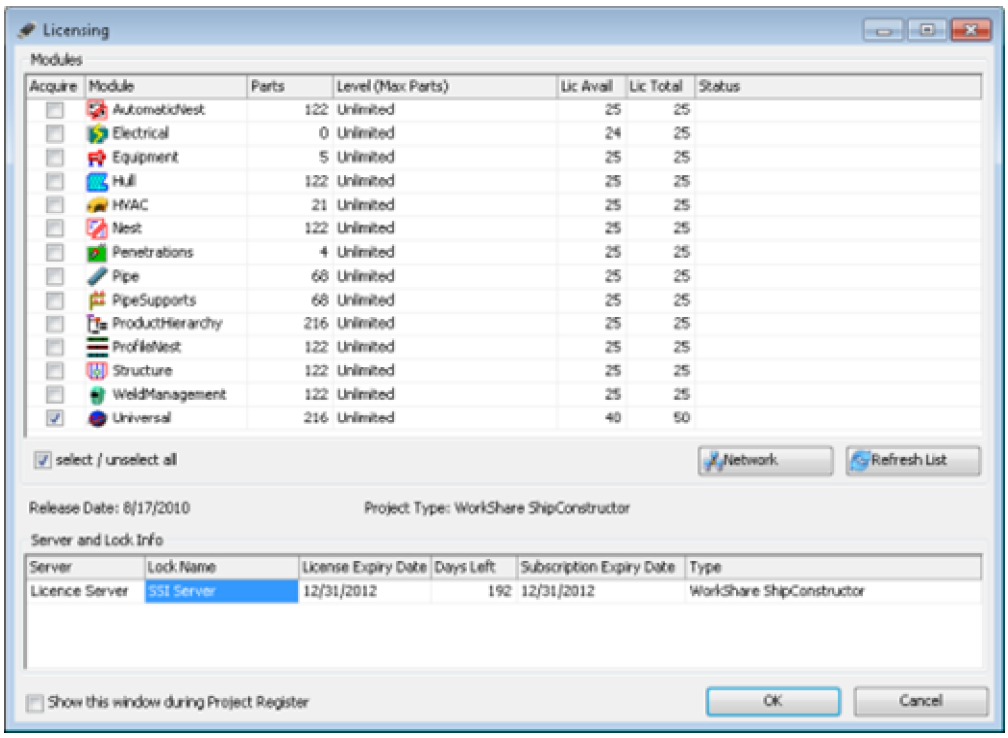
Get Licenses

You can turn on and off licenses to enable or disable ShipConstructor modules. You can purchase ShipConstructor modules separately at various levels (based on number of parts). You can also use a combination of local locks (located on your computer) and network locks (located on a network-accessible computer).

For details on setting up licenses, see the Installation Guide.

To turn on or off licenses

1. Choose ShipConstructor > Licensing (page 390) to open the Licensing window.



2. Set the options (see Licensing Window (page 390)).
3. Click OK.

Re-Login to ShipConstructor

If you have different user accounts for different roles, you may need to change who you are logged in to ShipConstructor as. To do so, re-open the project and enter a different user name and password (see Open a Project (page 11)).

Reload a Project Database

Whenever you or anyone else makes changes to a project in Manager, you must reload the project database for the change to appear in your ShipConstructor session.

To reload the project database

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.
2. Select the project node in the component tree on the left side of Navigator (for example, SC2009Demo).
3. Click Reload DB.

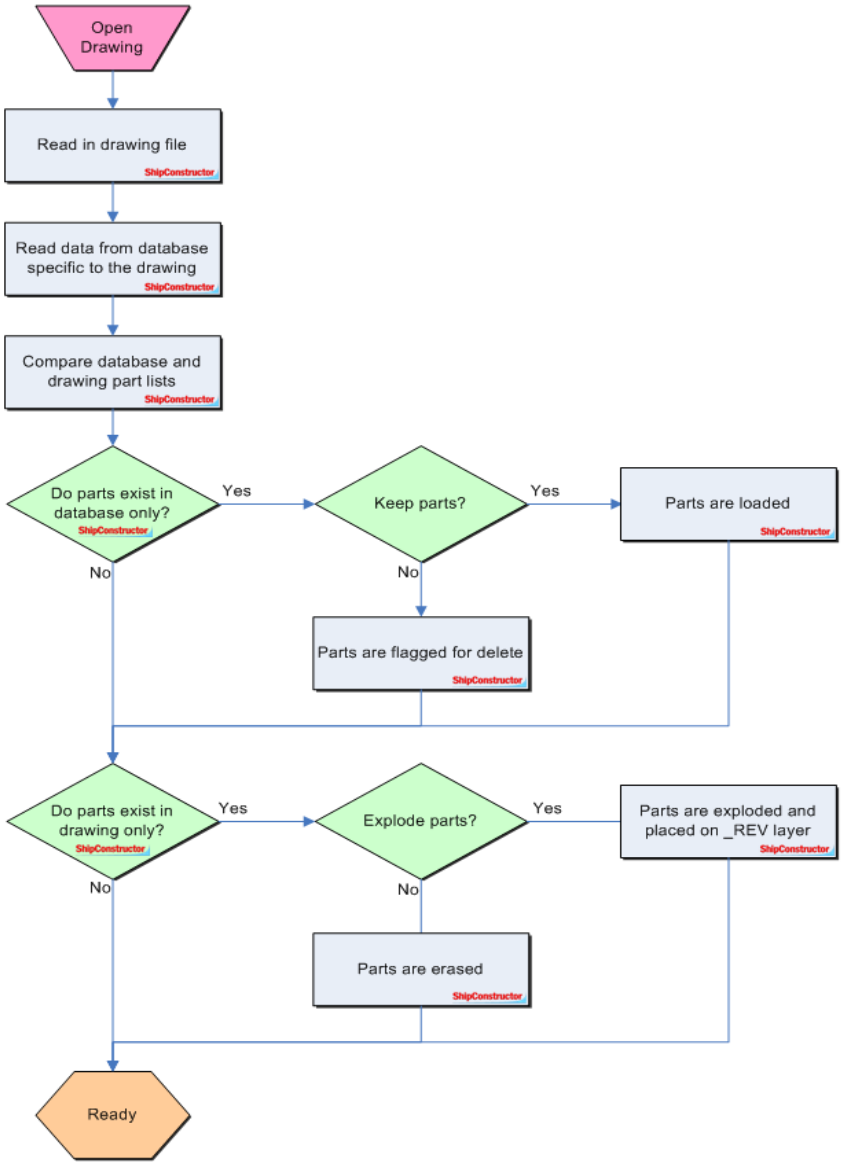
Change Your Password

User passwords are configured using the Administrator utility. Please consult the section Change User Passwords in the Project Management manual.

Drawings

Drawings give you access to the project model data. ShipConstructor saves the parts in the drawings but actually stores the parts in the database. If there is any discrepancy between the drawing and the database, the database is considered to be correct.

Note: ShipConstructor stores AutoCAD objects in the drawings only (not in the database).



Open a Drawing

To open a drawing

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.

Note: To open a drawing from a unit other than the current unit, select the project at the top of the component list, then select the unit in the Unit list.

2. Select the type of drawing in the component list (for example, Structure or Pipe).

   Note: To open a unit drawing, select the unit name in the component list (for example, U12).

3. Select the drawing in the drawing list.

   Note: To open the drawing in read-only mode, click on the Read Only check box.

4. Click Open.

Create a Drawing

To create a drawing

Note: The following procedure describes how to create drawings in general. For information on creating specific types of drawing, see the appropriate section of this manual.

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.

   Note: To create a drawing for a unit other than the current unit, select the project at the top of the component list (for example, SC2009Demo), then select the unit in the Unit list.

2. Select the type of drawing in the component list (for example, Structure or Pipe).

   Note: If the drawing list contains folders, select the folder for the sub-type of drawing.

3. Click New.

   Note: The New button label may be different for different drawing types (for example, New Piping, New Arrangement, Create, or Create Spool Dwg).

   The New Drawing window appears.

4. Type a File Name for the drawing.

   Note: To open the new drawing, click on the Open new drawing check box.

5. Click OK.

Lock or Unlock a Drawing

You can lock drawings so others can open them and view them but not save any changes to the drawing.

To lock or unlock a drawing

1. In Navigator, click the padlock icon beside the drawing name.
   - A dark padlock icon means the drawing is locked.
   - A dimmed padlock icon means the drawing is unlocked.

   Note: If clicking the lock icon does not seem to work, you may not have permissions to lock or unlock the appropriate drawing files.

Rename a Drawing

To rename a drawing

1. In Navigator, right-click on the drawing name and choose Rename.
2. Enter a new name.
3. Press Enter.

Delete a Drawing

To delete a drawing

1. In Navigator, right-click on the drawing name and choose Delete.

Save a Drawing

It is important for the user to click File > Save in AutoCAD to save the current (structural) drawing periodically. This ensures that the drawing and the project database remain synchronized.

> Note: The automatic saving feature through AutoCAD will not save ShipConstructor drawings.

Closing a Drawing

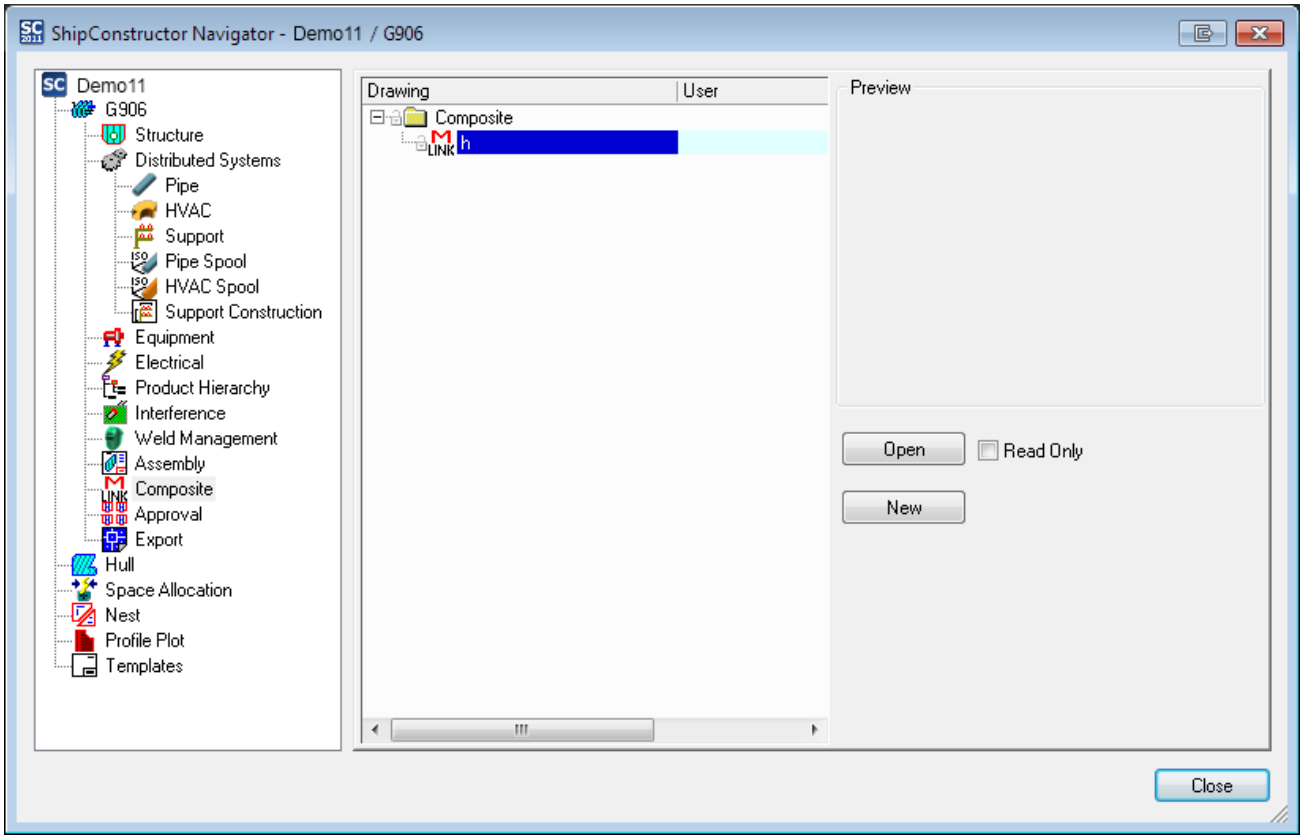
Erased parts are deleted from the database when the drawing is closed and you have saved the drawing. If AutoCAD crashes without closing a drawing, any parts that were deleted in that session will be left in the database. The next time the drawing is opened, a synchronize warning will ask you to choose to keep the objects that are only in the database.

Set Up Composite Drawings

You can create composite drawings in ShipConstructor and use AutoCAD's **XCLIP** command to create composite deck or frame drawings. Composite drawings make the design process easier and let you see exactly what is happening around a deck or specific frame, including the structure, pipe, equipment, and HVAC components, without making any permanent changes to your drawings.

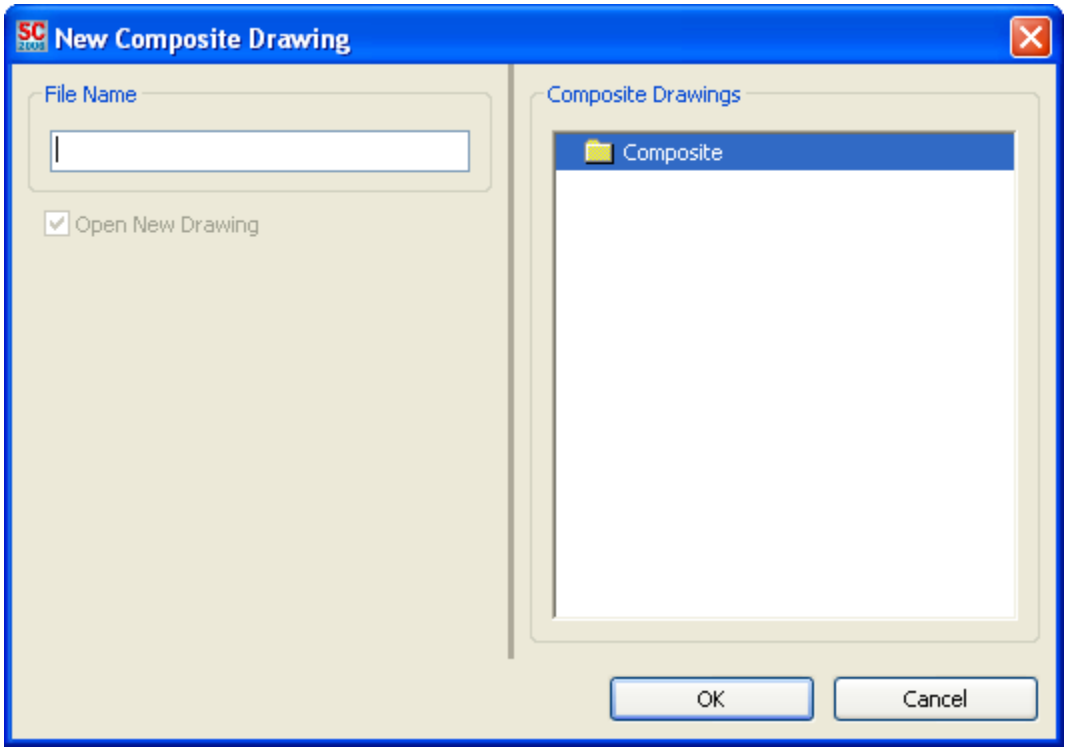
To create a composite drawing

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.



2. Choose Composite in the component list.

3. Click New.



4. Enter a File Name for the drawing.
5. Click OK.

   The Mlink Manager window appears.
6. Select the drawings you want displayed in the composite drawing.
7. Click OK.

Insert (Link) a Drawing into Another Drawing

Each drawing contains a relatively small portion of the entire project. Sometimes you want to see more of the project than is contained in a single drawing. To do so, you can insert (link) other drawings into the current drawing. This type of link is referred to as a model link or mlink. When you mlink other drawings into the current drawing, the objects in those drawings are visible in the current drawing. However, you cannot modify objects that are mlinked into the current drawing.

For example, if you are modeling structural components, you may want to see other structural elements, equipment, or piping by mlinking them into the current planar group drawing. Similarly, if you are modeling piping, you may want to see a few key structural drawings by mlinking them into the current pipe drawing. Mlinking lets you easily see the spatial relationships between drawings.

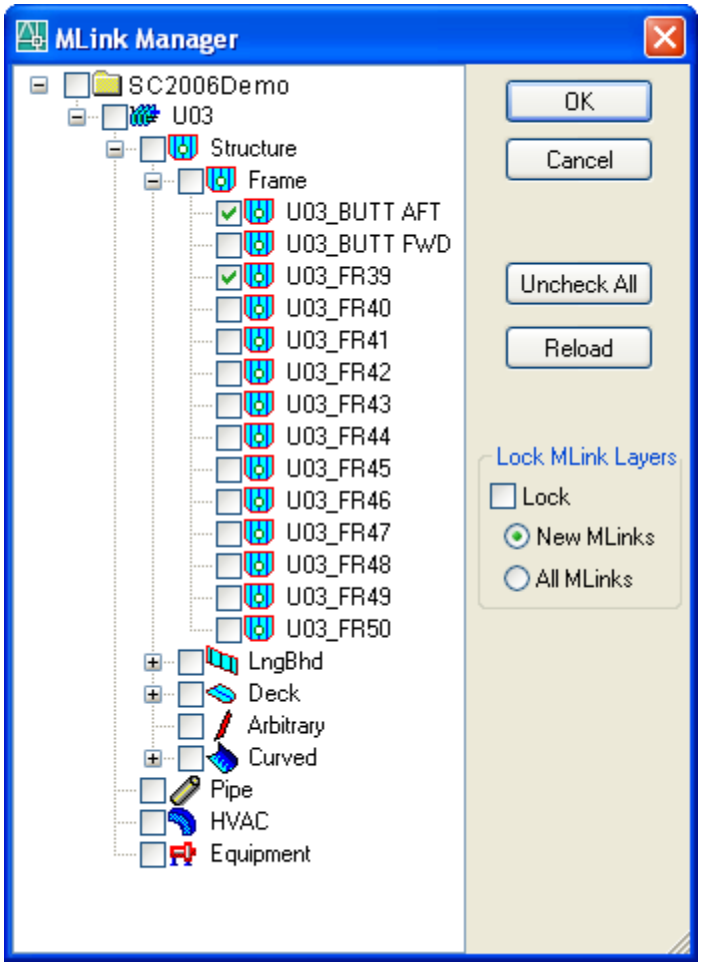
You can also use mlinks to control which drawings are included in a unit drawing. In some cases, it may take a long time for ShipConstructor to open the unit drawing because you have very complex planar group drawings. If so, you may want to temporarily remove certain mlinked drawings to allow the unit drawing to load faster. In other cases, you may want to use mlinks to include pipe and equipment in the unit drawing.

You can also use mlinks to create a composite drawing of the entire vessel (see Set Up Composite Drawings (page 16)).

Note: It can take several minutes for ShipConstructor to mlink all unit drawings into a composite drawing. (For example, it can take over thirty minutes to mlink all nine units of a reasonably sized project and pipe and equipment into a composite drawing.)

To mlink drawings into the current drawing

1. Choose ShipConstructor > Model Link (page 383) to open the MLink Manager.



2. Select the drawings to insert (link) into the current drawing.
3. Click OK.

> Tip: After inserting (linking) a drawing into another drawing, you can use AutoCAD's **XCLIP** command to hide parts of the linked drawing. For details, see xclip in AutoCAD Help.

To reload mlinked drawings

1. Choose ShipConstructor > Model Link (page 383) to open the MLink Manager.
2. Click Reload.

To remove an mlinked drawing

1. Choose ShipConstructor > Model Link (page 383) to open the MLink Manager.
2. Remove the checkmark from the drawings to remove the drawing from the current mlinked drawing.

> Tip: To remove all mlinked drawings, click Uncheck All.

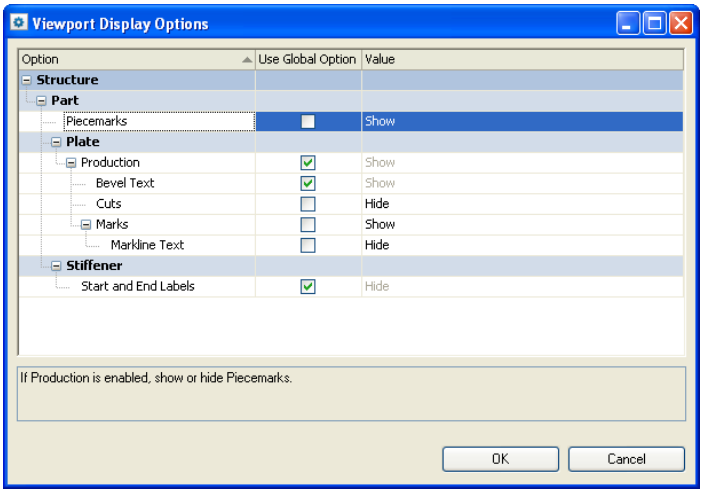
3. Click OK.

Configuring Viewport Dependant Display Options

Viewports in ShipConstructor can be configured with a subset of the options available globally. These selections will only show inside the specified viewport. This can be done inside of a Template or a production drawing.

Configuration changes made to the Viewport Display Options will override those same settings in the global display options. Each option will default to using the global option, even if other options are overrode unless the Use Global Option check box is unchecked the global option for that value will be used. If the Use Global Value option is unchecked then the value of that row will be used to determine the display settings for that viewport.

1. Enter the command SCVIEWPORTDISPOPTIONS on the command line.
2. If you were previously inside of a model space viewport the options for that viewport will be displayed, else you will be prompted to select the viewport that you wish to configure.

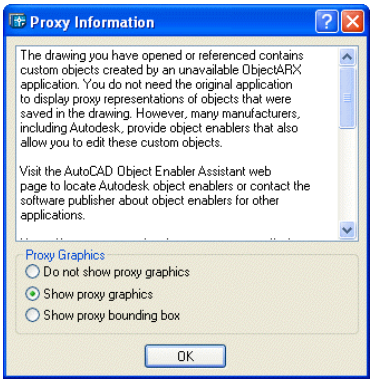


3. Once a viewport has been selected the Viewport Display Options window will appear. The option grid has three columns.
    - Option: The names of the different options as well as categories under which the options are organized.
    - Use Global Option: If checked then the global display options will be used instead of these options, if unchecked then the viewport specific option will be used.
    - Value: The viewport specific value for this option. This value will only be used if the Use Global Option is unchecked.

    Below the grid is the Description field, it will contain a description of the selected option.
4. Once configuration is done click **OK** to continue.

Exporting ShipConstructor Drawings

ShipConstructor drawings are similar to standard AutoCAD drawings, except they contain many custom objects. If you are running AutoCAD (without ShipConstructor) and try to open a ShipConstructor drawing, the following message appears.



You can install the ShipConstructor Object Enabler (page 23), a free download from www.ShipConstructor.com, that lets you open ShipConstructor drawings in AutoCAD.

You may also generate a document from your original ShipConstructor drawings that can be opened without ShipConstructor installed or the ShipConstructor Object Enabler (page 23) there are a number of methods depending on what you wish to produce.

- Export To Dwg: A function to export a single ShipConstructor drawing.
- Export Documents: An Exported Document that includes the entities from one or more original ShipConstructor drawings.
- Bound Approval Drawings: An Approval Drawing that has been exported into native format to avoid modifications to the source drawings modifying it and to allow it to be viewed independent of ShipConstructor.

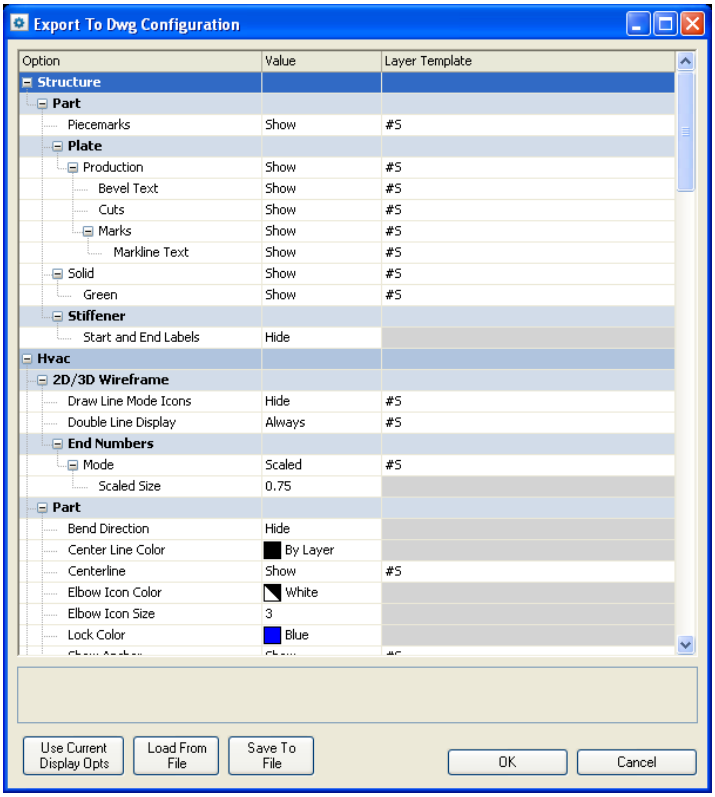
Export To Dwg

The Export To Dwg feature provides you with the ability to export a ShipConstructor document to a native AutoCAD document that can be viewed without the use of ShipConstructor or the ShipConstructor Object Enabler. The user is given the ability to select what elements of a drawing are exported and separate the elements of complex ShipConstructor parts onto separate layers in the new document.

Any parts that are MLinked into the drawing being exported will have local copies created out of native AutoCAD entities using the same settings to determine what elements of parts are included and the layer they are placed upon.

1. Choose SC Utilities > Export > Export To Dwg... (page 466) or type SCEXPORTDWG at the command line

   If ShipConstructor thinks the drawing may have been modified you will be asked if you wish to save the current drawing, the drawing must be saved for an export to proceeded.
2. A Save As window will appear with a generated name for the document. Set the file name and location as needed. Click **Save** and the Export To Dwg Configuration window will appear.



3. The Export To Dwg Configuration: The interface to configure the export will display the available options in the main grid. Items that are displayed under another item are dependent upon the item above them to be enabled for them to be enabled.

Example: Structure>Part>Solid>Green is set to Show in the above picture, but if Structure>Part>Solid was set to Hide then any dependant values, such as Green, would be be treated as having the value Hide and the tree under Solid would be collapsed for easier navigation.

Note: The configuration options for each module are discussed in the module's display option documentation.

Layer Templates

The Layer Template feature allows you to separate elements of the drawing out onto separate layers based on what option it is related to and the original layer it was placed upon.

The default setting for a layer template, <Source Layer>, means that that element will be left on exactly the same layer as it is in the original drawing.

If you wish to create an exported document that has all of the Structure solids on a layer called _SOLIDS then replace the layer template for Structure>Plate>Solids with _SOLIDS. If the layer does not already exist a new layer with that name will be created and all solids from structure elements will be placed upon it. Note that while the Stiffener Green is also a solid it is dealt with separately and if you wish it to be on the _SOLIDS layer it will also need to have the same Layer Template. You may place as many or as few options on the same layer as you wish.

Example: If an entity on layer _REV is exploded with the template "_SCEntity<Source Layer>" it will be placed on the layer "_SCEntity _REV"

Note: <Source Layer> is a special token in a Layer Template, it refers to the Source Layer and it will be replaced with the name of the source layer.

The buttons on the bottom left of the window are used to manage configurations.

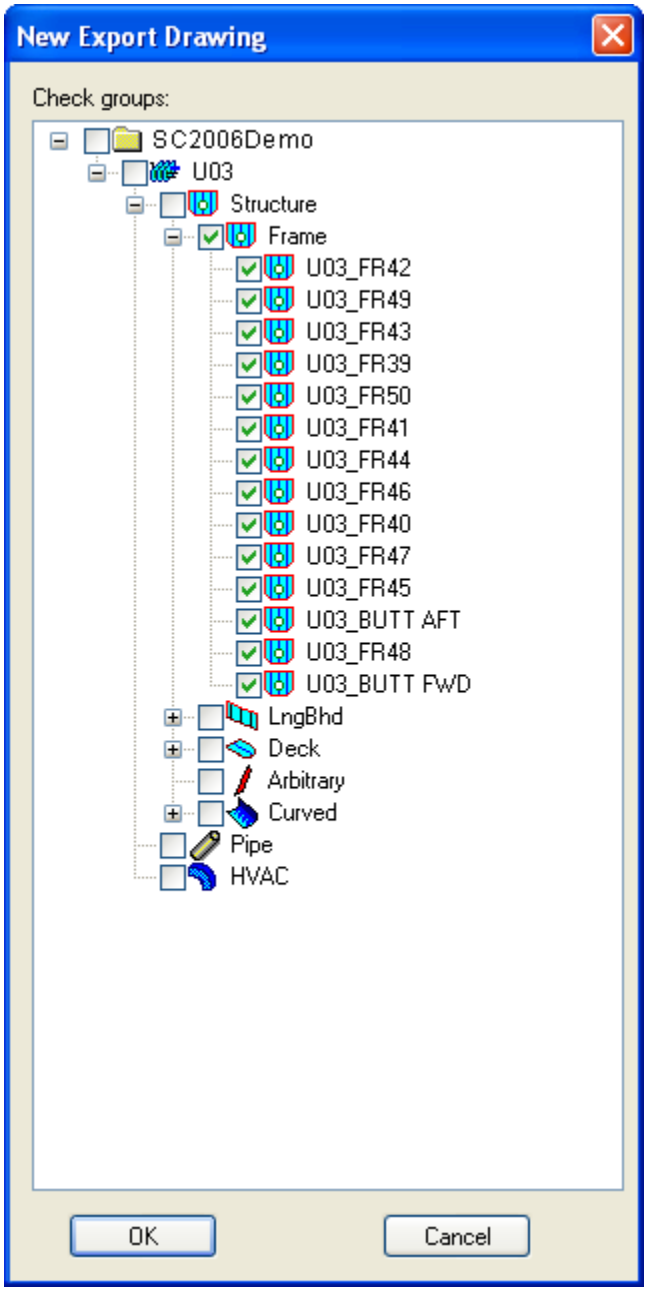
- **Use Current Display Opts** will read the current global display options from the source Approval Document and configure the UI accordingly.
- **Load From File** allows you to read in an previous Bound Approval Document Configuration and configure the UI accordingly.
- **Save to File** allows you to save the current settings in the UI to a text file that can be used to configure other Bound Approval Document configurations.

Export Drawings

ShipConstructor drawings can be exported to standard AutoCAD drawing format. This lets you use the solid model in other applications that do not support the object enabler. During export, ShipConstructor converts all custom objects to standard AutoCAD objects. Structure parts are exported as 3D solids and production geometry. In the standard AutoCAD drawing, exported solids are on the DrawingName_SOLID layer and production geometry is on the DrawingName_PRODUCTION layer.

To export ShipConstructor drawings

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.
2. Select Export in the component list.
3. Click New to open the New Export Drawing window.



4. Click the check box for the drawings to include in the **EXPORTED** drawing.
5. Click OK.

Object Enabler

You will not be able to open ShipConstructor drawings in plain AutoCAD without ShipConstructor or the ShipConstructor Object Enabler. The object enabler is a separate installation that contains a portion of ShipConstructor that allows the custom objects to be accessible. The object enabler contains no commands and is not connected to the project database. You cannot edit parts or construction lines. Changes or additions to non-ShipConstructor objects are permitted.

Note: Object Enabler is available as a free download from www.ShipConstructor.com. Do not install both the object enabler and ShipConstructor on the same computer.

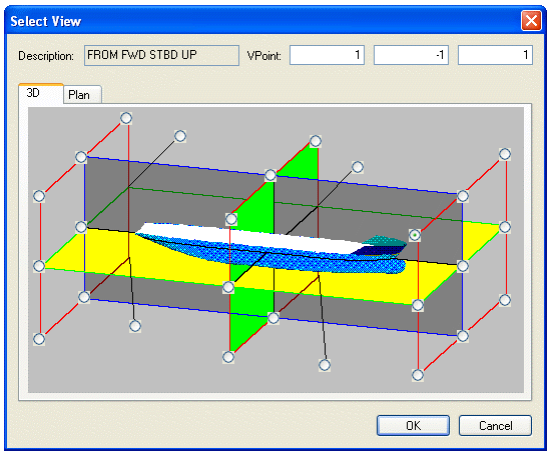
Views

You can adjust the view within any drawing.

Select a 3D or 2D View

To select a 3D or 2D view

1. Choose SC Utilities > 3D Viewpoint (page 451) to open the Select View window.
2. Select the 3D tab.



3. Select a point within the 3D ship hull image.

   Note: You can also manually enter the viewpoint values beside VPoint.

4. Click OK.

Switch Between Paper Space and Model Space

Each drawing contains two types of views.

- The model space view represents your model in three-dimensions. You use the model space view for modeling and drafting.
- The paper space view represents your model in two-dimensions as it would be printed or plotted on paper. You can use the paper space view for creating a finished layout for printing or plotting.

To switch between paper space and model space

1. At the bottom of the drawing window, click the Model tab to switch to model space or click the Layout tab to switch to paper space.

   Note: A drawing can have more than one Layout tab.

Windows and Editors

When a window or editor opens, there are numerous features designed to help you accomplish the task you are doing. Some of the most commonly used features are the following:

- If information is sorted in a column; you can click the column name to sort the list alphanumerically.
- Click the column name a second time to sort the information in reverse alphanumerical order.
- If a window consists of more than one tabbed panel, you can move between tabs using the left and right arrow keys (only when a tab name is highlighted).
- Press Tab to navigate among the window's elements.
- Press Esc to close a window.

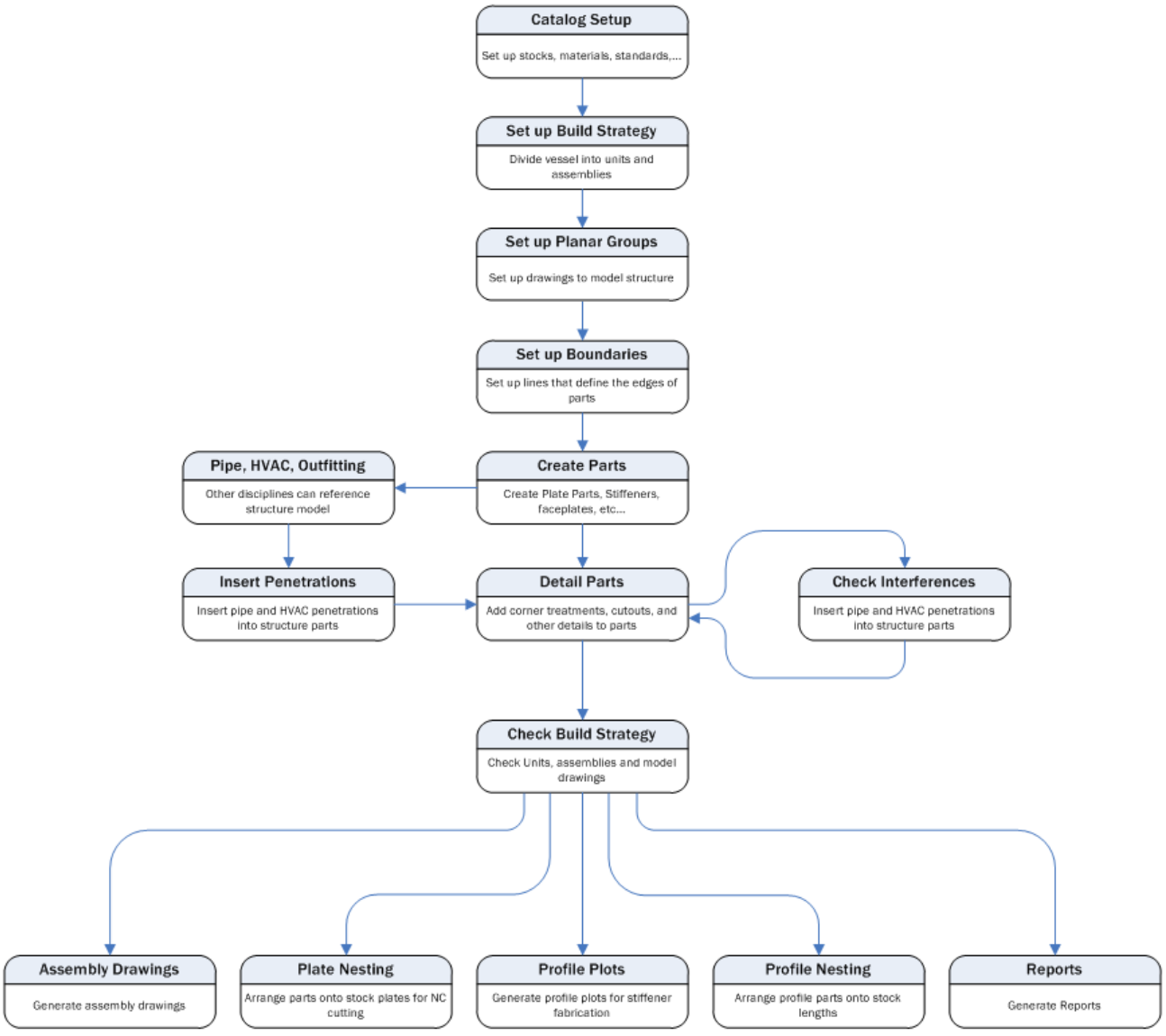
Structure

The Structure manual is organized to lead you through steps to set up a project.

Note: Before you can work in ShipConstructor, you must create a project. Typically, the project manager creates the ShipConstructor project. See the Project Management manual for details on project administration.

Structural Design Workflow

The workflow of ShipConstructor will differ between companies due to the specific work processes of the company. The following example is a typical high-level flowchart of the Structure discipline.


Catalog Setup
Set up stocks, materials, standards,...
Set up Build Strategy
Divide vessel into units and assemblies
Set up Planar Groups
Set up drawings to model structure
Set up Boundaries
Set up lines that define the edges of parts
Pipe, HVAC, Outfitting
Other disciplines can reference structure model
Create Parts
Create Plate Parts, Stiffeners, faceplates, etc...
Insert Penetrations
Insert pipe and HVAC penetrations into structure parts
Detail Parts
Add corner treatments, cutouts, and other details to parts
Check Interferences
Insert pipe and HVAC penetrations into structure parts
Check Build Strategy
Check Units, assemblies and model drawings
Assembly Drawings
Generate assembly drawings
Plate Nesting
Arrange parts onto stock plates for NC cutting
Profile Plots
Generate profile plots for stiffener fabrication
Profile Nesting
Arrange profile parts onto stock lengths
Reports
Generate Reports

Structure Library Setup

Overview

Each project contains various libraries with all the stocks, materials, and standards that you will use in the project. Before you begin work on a project, you must set up these project libraries. You can also import project libraries from another project or from an XML file that you have exported from another project. Virtually all the settings can be changed after parts have been modeled.

This manual describes how to set up general libraries and structure libraries. See the Pipe manual for details on setting up pipe libraries, the Equipment and Penetrations manual for setting up equipment libraries and penetration standards, and the HVAC manual for details on setting up HVAC libraries.

Supplied Libraries

ShipConstructor comes with three template XML files that contain sample standards and stocks. By default these files are found at C:\Program Files\ShipConstructor2009\ProjectTemplates.

- MetricTemplate.XML – metric stocks and standards
- ImperialTemplate.XML – imperial stocks and standards
- PipeCatalog.XML – pipe catalogs

Catalog Setup Workflow

This manual is organized in the order that you will typically set up a project.

Project Settings

There are certain settings that ShipConstructor uses throughout a project. Before you begin work on a project, you must set these project settings. We recommend that you do not modify certain project settings after you have begun work on a project. See General > Project Settings (page 327) for details.

You can also import project settings from another project or from an XML file that you have exported from another project. We recommend that you do not modify the project settings after you have begun work on a project.

Set Project Settings

To set the structure options

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Project Settings (page 327) to open the Project Settings window.
3. Open the Structure section and set the options. See General > Project Settings (page 327).
4. Click OK to close the Project Settings window.

Import Project Settings

To import project settings

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Project Settings (page 327) to open the Project Settings window.
3. Click Import to open a File Browser.
4. Select a project file (*.PRO) or XML file (*.XML).
5. Click Open.

Export Project Settings

To export project settings

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Project Settings (page 327) to open the Project Settings window.
3. Click Export XML to open a File Browser.
4. Enter a name for the XML file.
5. Click Save.

User Permissions

See the Project Management manual for details on how to use User Permissions.

ShipConstructor is designed to be used by a team of people. Each company will have different types of people, so we will present a typical case.

Structure Team Roles

Project Manager:

- Schedules and plans the project
- Organizes team
- Corresponds with client

Structural Designer:

- Approves contract drawings
- Creates structural detail design

Structural Lead:

- Ensures that standards are maintained
- Is responsible for the model

Build Strategist:

- Develops the build strategy for the project
- Coordinates scheduling

Modeler:

- Models all structure and generates parts

Nester:

- Creates nest and nests parts
- Tracks stock

Production Detailer:

- Details assembly drawings

CAD Administrator:

- Creates users and maintains databases
- Manages communication about issues with ShipConstructor Software Inc. and applies patches

People in the team may have more than one of these roles. The different roles in the project require different permissions. We recommend that you set up permission groups for each role.

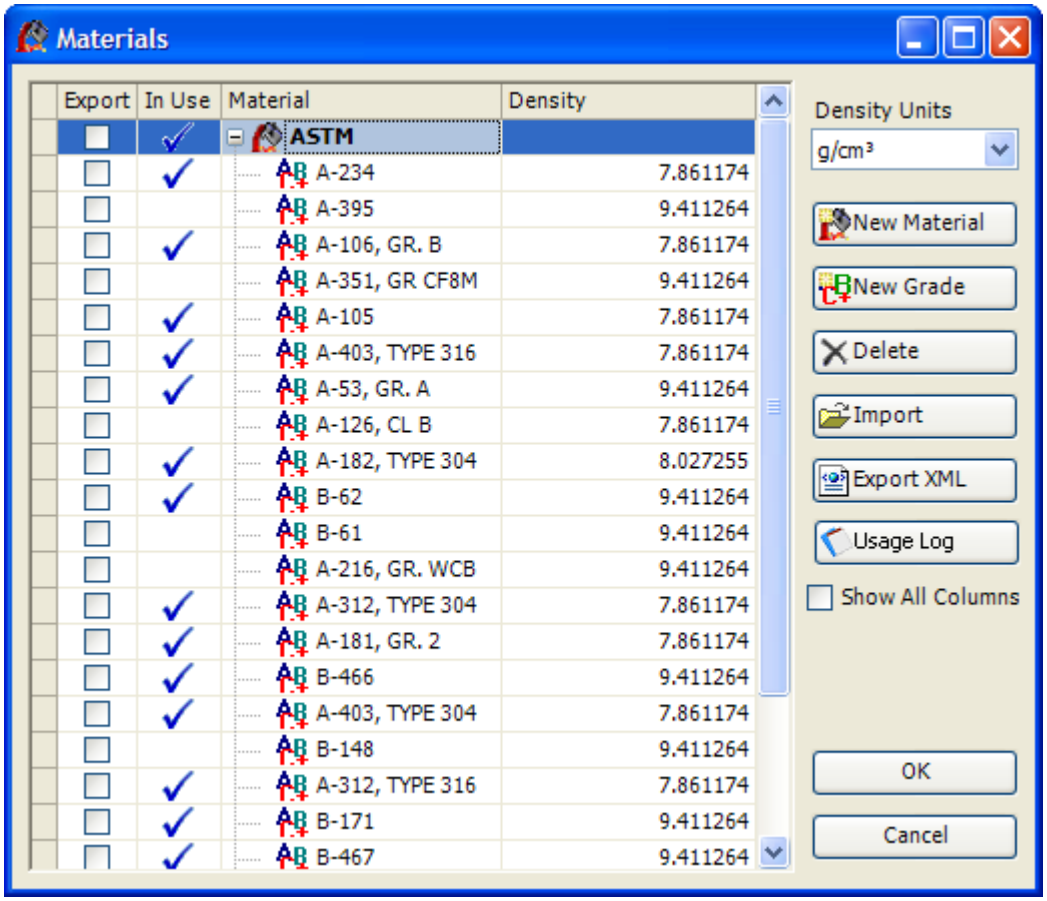
Materials

Each part is fabricated from a specific type of material (for example, Aluminum 5086). You must set up the materials you plan to use in the project. In ShipConstructor, a material is actually a grouping of material grades. For example, the material Aluminum might contain the grades 5086, 6061, and 5052.

Create a Material

To create a material

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Materials (page 332) to open the Materials window.



3. Click New Material.

4. Type a name for the material and press Enter.

Create a Grade

To create a grade

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Materials (page 332) to open the Materials window.
3. Select the Material in which to create a grade.
4. Click New Grade.
5. Enter a name for the grade and press Enter.
6. Make sure Density Units is set to the units you want to use to specify the density.
7. Double-click the Density field for the new grade, enter a value (measured in the Density Units), and press Enter.
   ShipConstructor uses density values to calculate part weight.

Rename a Material or Grade

To rename a grade

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Materials (page 332) to open the Materials window.
3. Double-click the material or grade to rename (or right-click the material or grade and choose Rename).
4. Enter a new name for the material or grade and press Enter.

Import Materials and Grades

To import materials and grades

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Materials (page 332) to open the Materials window.
3. Click Import to open a File Browser.
4. Select a project file (*.PRO) or XML file (*.XML).
5. Click Open.
6. Check the materials in the Import column that you want to import and click OK.

Export Materials and Grades

To export materials and grades

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Materials (page 332) to open the Materials window.
3. Check the Export check boxes for the materials and grades to export.
4. Click Export XML to open a File Browser.
5. Enter a name for the XML file.
6. Click Save.

Delete a Material or Grade

To delete a material or grade

> Note: You cannot delete a material or grade that is currently in use in the project. You can produce a log of all stocks/standards that use a particular material grade by selecting the material grade which is checked “In Use”, and clicking Usage Log.

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Materials (page 332) to open the Materials window.
3. Select the material or grade.
4. Click Delete.

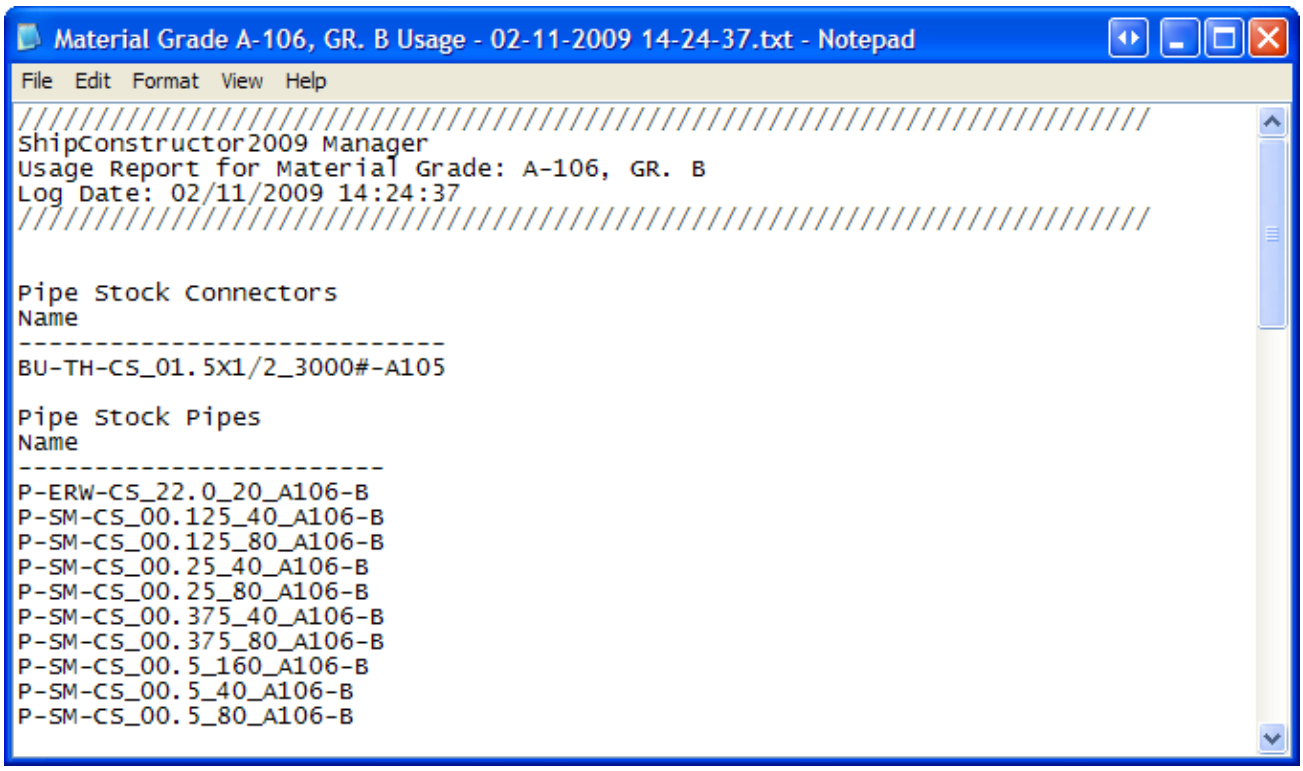
View Material Usage

It is possible to see a list of all the stocks that use a selected material grade. A check mark in the In Use column indicates that the material grade is being used by stocks.

To list all stocks using a specific grade

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Materials (page 332) to open the Materials window.
3. Select the grade you want.
4. Click Usage Log.

ShipConstructor displays a list of all stocks using the selected grade.



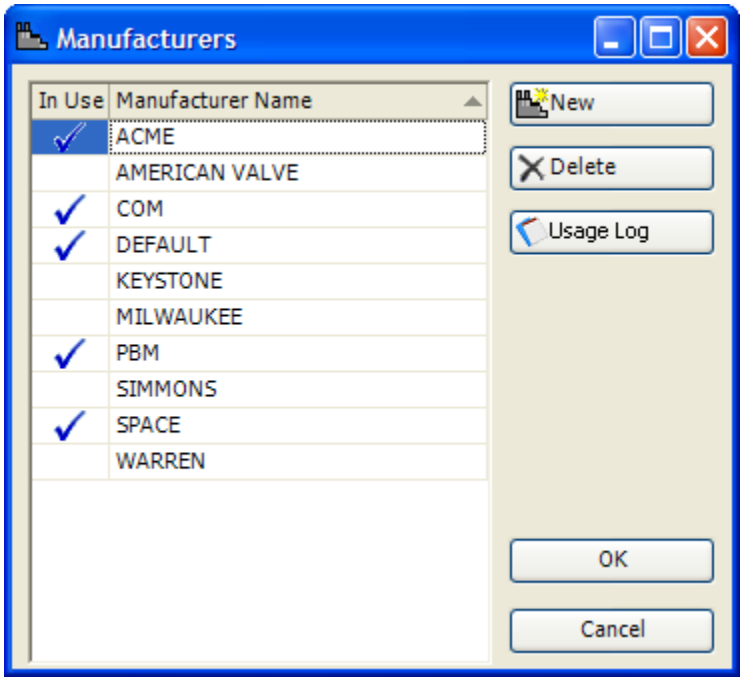
Manufacturers

A manufacturer is a company that manufactures certain items used in your project (for example, pipe stock and insulation). When you create such an item, you must assign it a manufacturer.

Create a Manufacturer

To create a manufacturer

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Manufacturers (page 333) to open the Manufacturers window.



3. Click New to create a new manufacturer.
4. Enter a name for the manufacturer and press Enter.

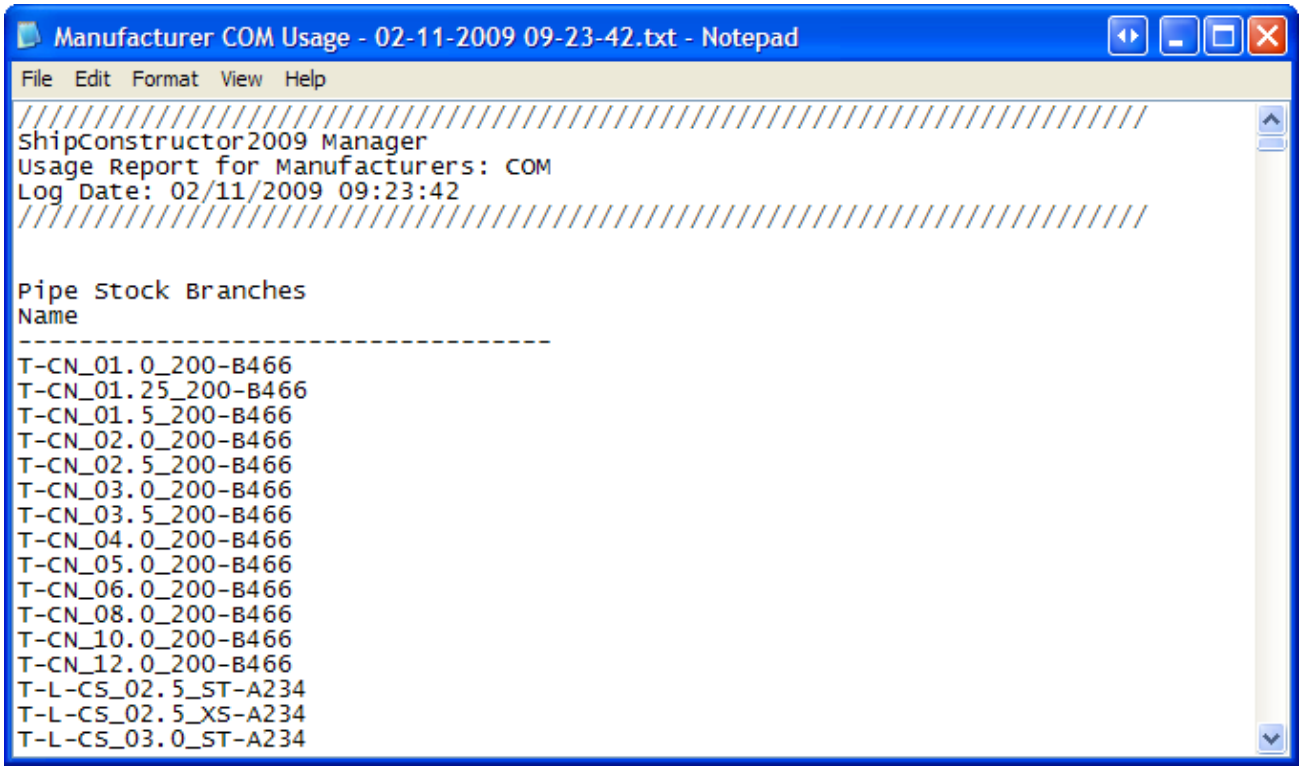
View Manufacturers Usage

It is possible to see a list of all the stocks that use a selected manufacturer. A check mark in the In Use column indicates that the manufacturer is being used by stocks.

To list all stocks using a specific manufacturer

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Manufacturers (page 333) to open the Manufacturers window.
3. Select the manufacturer you want.
4. Click Usage Log.

   ShipConstructor displays a list of all stocks using the selected manufacturer.



Finishes

Parts often have a finish applied to them (for example, paint). You must set up the finishes you plan to use in the project. A finish type is a group where you can set up similar types of finishes. For example, you may have a finish type named Paint that contains finishes like antifouling, primer, and enamel.

Create a Finish Type

To create a finish type

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Finishes (page 334) to open the Finishes window.
3. Click New Type.
4. Type a name for the finish type and press Enter.

Create a Finish

To create a finish

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Finishes (page 334) to open the Finishes window.
3. Select the finish type that the new finish will belong to.
4. Click New Finish.
5. Type a name for the finish and press Enter.

Rename a Finish or Finish Type

To rename a finish or finish type

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Finishes (page 334) to open the Finishes window.
3. Double-click the finish or finish type to rename (or right-click the finish or finish type and choose Rename).
4. Type a new name and press Enter.

Import Finishes and Finish Types

To import finishes

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Finishes (page 334) to open the Finishes window.
3. Click Import to open a File Browser.
4. Select a project file (*.PRO) or XML file (*.XML).
5. Click Open.

Export Finishes and Finish Types

To export finishes

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Finishes (page 334) to open the Finishes window.
3. Check the Export check boxes for the finishes and finish types to export.
4. Click Export XML to open a File Browser.
5. Enter a name for the XML file.
6. Click Save.

Delete a Finish or Finish Type

To delete a finish or finish type

Note: You cannot delete a finish or finish type that is currently in use in the project.

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Finishes (page 334) to open the Finishes window.
3. Select the finish or finish type.
4. Click Delete.

View Finishes Usage

It is possible to see a list of all the parts that use a selected finish. A check mark in the In Use column indicates that the finish is being used by parts.

To list all parts using a specific finish

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Finishes (page 334) to open the Finishes window.
3. Select the finish you want.
4. Click Usage Log.

Insulation

Insulation is not used by the Structure module. See the Pipe manual for details on insulation.

Text Styles

Text styles let you control the appearance of plate part text (marklines and piecemarks) in production drawings.

To set up a text style, you must set up both a ShipConstructor text style name (which applies to the entire project) and an AutoCAD text style with the same name that defines the appearance of the text, which applies to an individual drawing. Whenever you insert text of a certain style, ShipConstructor formats it using the AutoCAD text style of the same name (or using the Standard style if there is not an AutoCAD text style with the same name).

For example, to standardize the appearance of piecemark text in assembly drawings, you would do the following. Set up a ShipConstructor text style named Piecemark and an AutoCAD text style named Piecemark in the assembly drawing template. When inserting text in any drawing, use the ShipConstructor text style Piecemark. You do not necessarily need to set up an AutoCAD text style named Piecemark in all drawings, since it is the appearance of text in the assembly drawing that you are trying to standardize. Conversely, different modelers can set up their own AutoCAD text styles named Piecemark in their own drawings. As long as they use the ShipConstructor text style Piecemark, all text of that style will be formatted consistently in the assembly drawings.

Create a Text Style

To create a text style

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Text Styles (page 335) to open the Text Styles window.
3. Click New.
4. Enter a name for the text style and press Enter.
5. Click OK to close the Text Styles window.

Delete a Text Style

To delete a text style

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Text Styles (page 335) to open the Text Styles window.
3. Select the text style you want to delete.
4. Click Delete.
5. Click OK to close the Text Styles window.

Accessory Packages

Accessory packages are not used by the Structure module. See the Pipe manual for details on accessory packages.

Naming Conventions

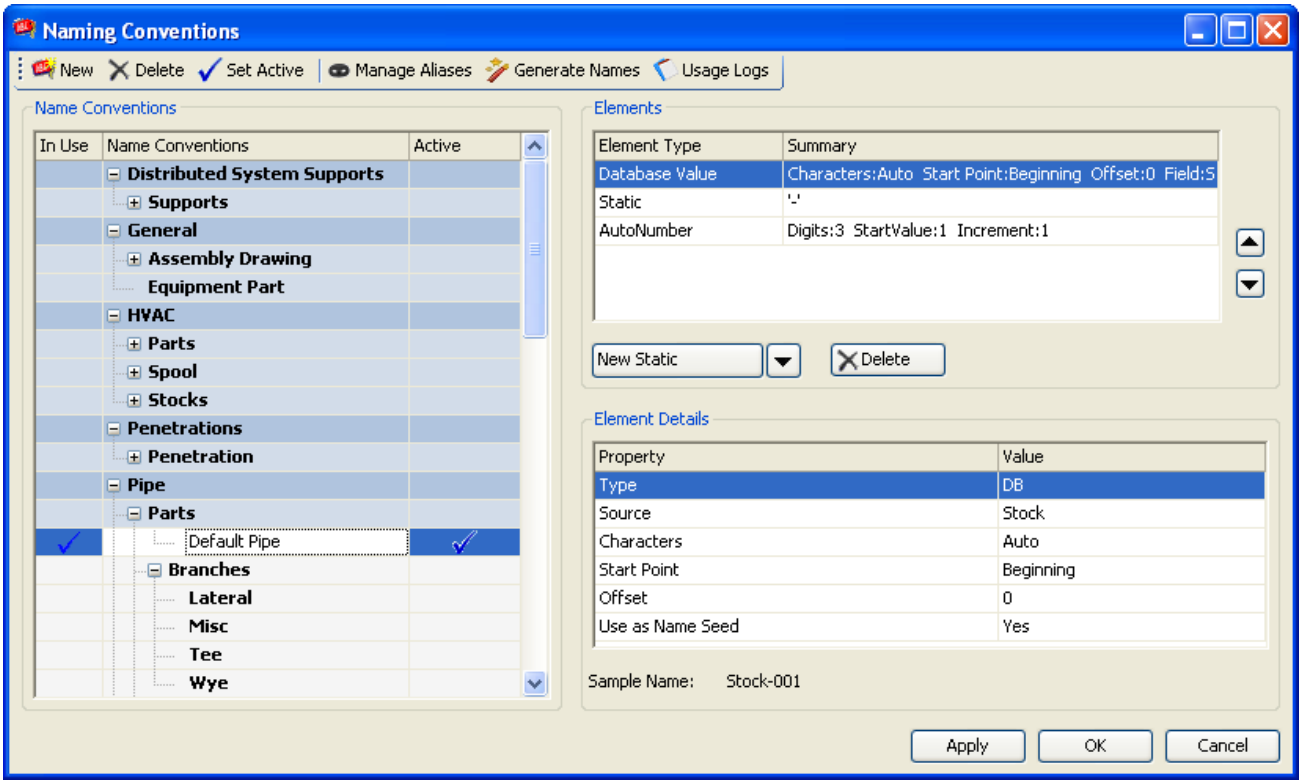
Naming conventions control how ShipConstructor automatically names elements of your project such as drawings, parts, spools, penetrations, and so on.

There are many types of naming conventions. ShipConstructor organizes them within a tree.

You can set up several different naming conventions within each category, but only one naming convention in each category can be the active naming convention that is used by ShipConstructor.

A naming convention is composed of elements. You set up a naming convention by choosing various elements, arranging them in order, and setting their values. When ShipConstructor automatically generates a name, it formats the name using the elements of the active naming convention.

Naming Conventions Window



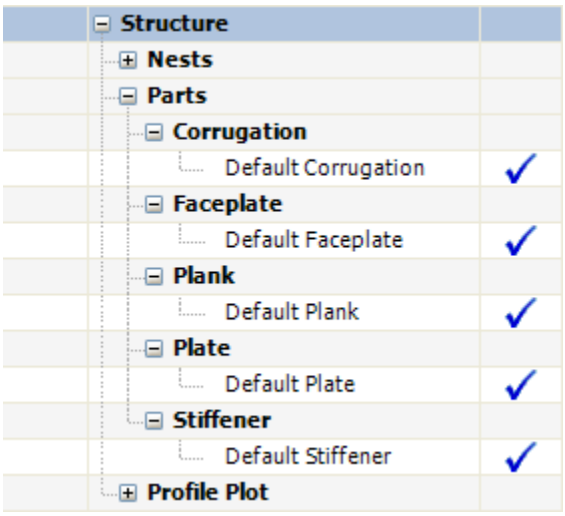
The Naming Conventions window consists of three information views.

- Naming Conventions tree (left)
- Naming Convention Elements list (top right)
- Element Details list (bottom)

Naming Conventions Tree

The Naming Conventions tree displays all the ShipConstructor entity types that names can be generated for and the naming conventions created for those entities. For each naming convention, ShipConstructor uses the Active column to show if the naming convention is the active naming convention for the entity type. Through the InUse column, ShipConstructor shows if there are any entities in the project that have a name generated with each naming convention.

Naming conventions can be defined at any level in the tree and apply to an entity type and all of its children entity types. For example, you define a naming convention PartNameConvention (convention A) for Structure > Parts and a naming convention PlateNameConvention (convention B) for Structure > Parts > Plate.



If you create a part of any type other than Structure > Parts > Plate, ShipConstructor will use naming convention A to generate the name. If you create a Structure > Parts > Plate, ShipConstructor will apply naming convention B.

You can define multiple naming conventions for each entity type. Depending on the type of entity, ShipConstructor may or may not allow you to pick a name convention to use when creating entities. In cases where you are not offered a choice, the Active naming convention is used.

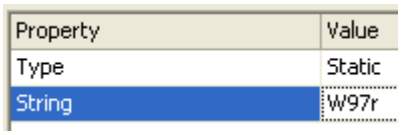
Naming Convention Elements List

A naming convention consists of elements that can be added, removed, and re-ordered. The elements list shows the elements of a naming convention and their order. The elements list shows element properties in the selected naming convention. Changing the properties of an element is done using the Details list.

There are three types of elements:

1. Static strings
2. Database items
3. Auto-Incrementing numbers

Static strings are user-entered text such as a word or a symbol (for example, a hyphen (-) to separate other elements).



Database items are a property of the entity for which the name is being generated. For example, the spec that an HVAC duct belongs to or the thickness of the plate stock that a plate part is made from. There are many different types of database items to choose from depending on the entity type that you are defining a naming convention for. The properties of a database item element are as follows:

- Number of characters – The maximum number of characters to display from the database item string.
- Start Point -- The position to start counting the number of characters from. The options are Beginning or End.
- Offset – The number of characters to skip from the offset before including the number of characters.
- Use As Name Seed – Indicates whether or not to use this element when determining the next auto-number choice. This will be explained in more detail below.

Property	Value
Type	DB
Source	System
Characters	Auto
Start Point	Beginning
Offset	0
Use as Name Seed	Yes
TreeLevel	Spec

Auto-Incrementing numbers are used to facilitate the creation of unique names. The properties of an auto-incrementing number are the following:

- Number of digits – A number representing the minimum number of digits used to display the auto-number. Can be set to Auto to automatically use as many digits as the number consists of. For example, if the generated number is 56 and the number of digits is set to 4, the auto-number would display as 0056. If the number of digits was set to Auto the number would display as 56.
- Start Value – The auto-number of the first generated name.
- Increment – The incremental value to add to each auto-number to calculate the next number. For example, an increment of 1 would create 6, 7, 8, 9, and an increment of 3 would create 6, 9, 12, 15.
- Recycle Numbers – A yes or no value. If you set the value to yes, then any entities that are renamed or deleted will have their name go back into a pool to be re-used again. For example, if you have recycling enabled and you create plate parts named P-67, P-68, and P-69 and then delete part P-68, the next plate part that you create will have the name P-68, not P-70.

Property	Value
Type	AutoNum
Digits	Auto
Start Value	1
Increment	1
Recycle Numbers	Yes

Element Details List

The Details list shows the properties of the element selected in the Elements list.

Create a Naming Convention

Before being able to add, edit, and remove elements, create and change aliases, and modify the naming convention itself, you must create your naming convention.

To create a naming convention

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Naming Conventions (page 336) to open the Naming Conventions window.
3. In the Name Conventions pane, select the type of naming convention to create. Creating a lower level naming convention will override one at a higher level.
4. Click New. ShipConstructor creates a new entry in the tree with a default name.
5. Type a name for the naming convention and press Enter.
6. Add elements to the naming convention. See Add an Element to a Naming Convention (page 38)

Add an Element to a Naming Convention

To add an element to a naming convention

1. Choose ShipConstructor > Manager (page 383) to open Manager.

2. Choose General > Naming Conventions (page 336) to open the Naming Conventions window.
3. In the Name Conventions pane, select the naming convention.

   Any existing elements are displayed in the Elements pane.

   Under Elements, a button with one of the labels described above allows you to add an element.

   If the label doesn't match the element you want to add, click to show the other types of elements.

   Note: Make sure each naming convention consists of at least one Static element and one AutoNumber element.
4. The new element is listed under Elements and is automatically selected.

   The element's properties are listed in the window.
5. Set the Value for each Property. See Edit an Element of a Naming Convention (page 39).
6. Click Test to generate a sample name.
7. If you are satisfied with your convention, click OK to close the Naming Conventions window.

Edit an Element of a Naming Convention

To edit an element of a naming convention

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Naming Conventions (page 336) to open the Naming Conventions window.
3. Under Name Conventions, select the naming convention.
4. Under Elements, select the element you want to edit.
5. Beneath the Elements pane, set the Value for each Property.
6. Click Test to generate a sample name.
7. If you are satisfied with your convention, click OK to close the Naming Conventions window.

Re-order Elements in a Naming Convention

To re-order elements in a naming convention

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Naming Conventions (page 336) to open the Naming Conventions window.
3. Under Name Conventions, select the naming convention.

   The naming convention's elements are listed in the Elements pane in order.
4. From the Elements pane, select the element and then click the up and down arrows to move the element up or down in the list
5. Click Test to generate a sample name.
6. If you are satisfied with the convention, click OK to close the Naming Conventions window.

Remove an Element from a Naming Convention

To remove an element from a naming convention

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Naming Conventions (page 336) to open the Naming Conventions window.
3. Under Name Conventions, select the naming convention.
4. Under Elements, select the element you want to remove.
5. Under Elements, click Delete.

Activate a Naming Convention

Because you can set up many naming conventions, you must choose one to be the active naming convention for a project.

To activate a naming convention

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Naming Conventions (page 336) to open the Naming Conventions window.
3. Under Name Conventions, select the naming convention you want to activate.
4. Click Set Active.

   A checkmark appears beside the naming convention, indicating that it is the active naming convention for that category.

Rename a Naming Convention

To rename a naming convention

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Naming Conventions (page 336) to open the Naming Conventions window.
3. Under Name Conventions, select the naming convention you want to rename.
4. Click Rename.
5. Enter a new name and press Enter.

Delete a Naming Convention

To delete a naming convention

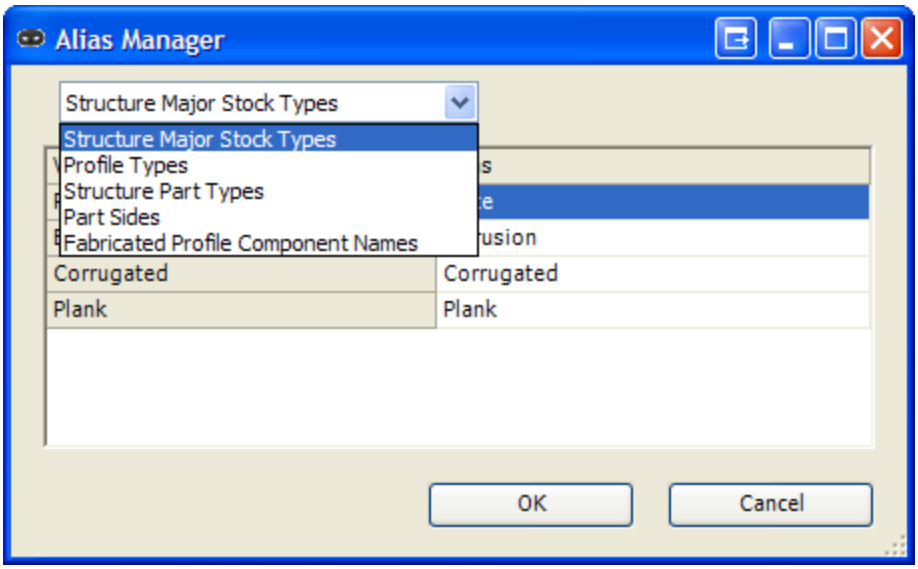
1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Naming Conventions (page 336) to open the Naming Conventions window.
3. Under Name Conventions, select the naming convention you want to delete.
4. Click Delete.

Managing Aliases

Aliases are used in naming conventions to customize the way that some database items are displayed. For example, when defining a naming convention for a structure plate part, you may include the part side in the name. By default the part sides are displayed as Port, Starboard, and Centerline. You may, however, want to shorten those to PT, ST, and C. This can be accomplished using naming convention aliases.

To start the alias manager

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Naming Conventions (page 336) to open the Naming Conventions window.
3. Click the Manage Aliases button on the bottom left on the Naming Conventions window to open the Alias Manager window.



To define an alias

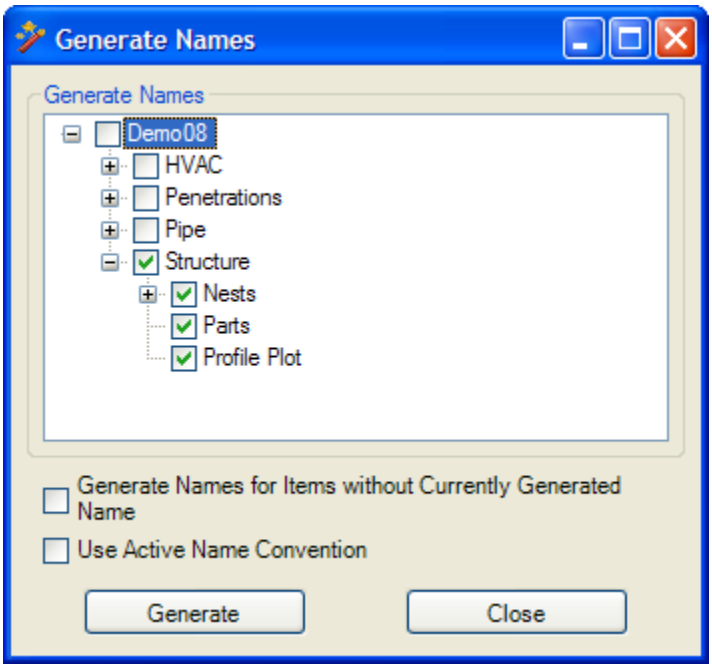
1. Open the Alias Manager.
2. Select the Database Item you wish to create an alias for from the drop-down box.
3. Double-click under the Alias column the row corresponding to the item you wish to create an alias for.
4. Type your alias and press the Enter key.
5. Click the OK button.

Generating Names in Groups

The Generate Names window is used for generating names for groups of items without having to open drawings.

To open the Generate Names window

1. Click the Generate Names button in the Naming Convention window.



The Generate Names window displays a hierarchy of ShipConstructor entities representing categories of nameable items in ShipConstructor.

To Regenerate Names for a Set of ShipConstructor Entities

1. Click the checkboxes in the tree corresponding to the ShipConstructor entities that you wish to regenerate names for
2. Click the Generate button

For all of the entities contained in the groups selected in the tree, ShipConstructor regenerates names using the Name Convention that was used to generate the original name. Any entities that do not have an existing generated name will not have names generated for them.

ShipConstructor also attempts to keep the same auto number for the regenerated names. For example, you change a naming convention to remove the part side element. The part whose name is Unit01-Port-436 will be renamed to Unit01-436 if that name is available. Otherwise, it will take the next available name. This feature is also beneficial because when regenerating names, if the naming convention definition has not been changed, parts that already have a generated name will not have their name changed.

There are two additional checkbox options on the Generate Names window.

- All Items
- Use Active Name Convention

Selecting the All Items option will generate names for all ShipConstructor entities that match the groups selected in the tree regardless of whether or not they have an existing generated name. If the Use Active Name Convention option is not set, entities that have an existing generated name have their names regenerated using the Name Convention their name was last generated with. Entities that do not have a generated name will have a name generated using the current Active Name Convention.

Selecting the Use Active Name Convention option will cause all generated names to be generated using the current active Name Convention for their category.

Naming Convention Concepts

Understanding Seeding

Seeding allows you more control over how your names are generated. This is best illustrated by an example. Suppose you have a naming convention defined with three elements: Material - System - Auto-number. The auto-number starting number is one and the increment is one. The first part you create may have a name like:

M1-FreshWater-001

In this example, the part is made from a material named M1 and is in the Fresh Water system. For this example we will assume that both the Material element and the System element are seeds. If you create a new part that was also made from material M1 and in system Fresh Water, the name will be as follows:

M1-FreshWater-002

If you create a third part that was made of material M1 but was in the Salt Water system, ShipConstructor will generate a part name as follows:

M1-SaltWater-001

Notice how the number has restarted at 1. This is because one of your seed elements has changed, in this case the System. If you create a fourth part, this time back in the Fresh Water system but with material M2, ShipConstructor would generate a name as follows:

M2-FreshWater-001

Again the numbering scheme has changed because the combination of your two seed elements does not match anything we have seen previously. If you now create a fifth and sixth part made from material M2 and in system Fresh Water, ShipConstructor will generate the names as follows:

M2-FreshWater-002

M2-FreshWater-003

To further illustrate the nature of seeding, consider the following example. To make things simple, pretend that you are again working with a fresh project and have not generated the names above. You define a name convention identical to

the one in the example above (Material - System - Auto-number) except that instead of both the Material and System elements being seeds, only the Material is a seed element. Following the same procedure as above your part names will look as follows:

M1-FreshWater-001

M1-FreshWater-002

M1-SaltWater-003

Notice how even though the system changed from Fresh Water to Salt Water, the numbering scheme did not change; it continued counting up. This is because the system is not a seed element. If you create two more parts, the names will be as follows:

M2-FreshWater-001

M2-SaltWater-002

Notice how the numbering scheme changed the first time because the material element changed but continued counting up the second time.

Database Elements

The power in naming conventions lies in the ability to generate names based on entity attributes stored in the database. The following are descriptions of all the database elements currently supported by ShipConstructor.

Assembly

Assembly returns the primary build strategy assembly name of an ancestor assembly of your entity. Which level's assembly name is returned depends on which Assembly Tree Level you select from the Tree Level drop-down list. The list of assembly levels depends on how you structure your Product Hierarchy tree.

> Note: If the part is in an assembly with a level higher than selected assembly tree level, nothing is returned for that element.

Extrusion Profile Standard Type

Extrusion Profile Standard Type applies to stiffeners. Examples of extrusion profile standard types are angle, bulb flat, structural pipe, custom, and tee. Extrusion profile standard types can be aliased using the Naming Convention Alias Manager.

File Path

File Path applies to Plate Nests. It returns the path to the plate nest drawing starting from the Nest folder. The purpose of the file path element is to allow users to put the build strategy location in the plate nest name.

This is done by creating a directory structure in the Nests folder that matches the build strategy and then putting plate nest drawings at the appropriate location in the folder structure.

Major Stock Type

Major Stock Type applies to all Structure parts. Examples of major stock types are plate, extrusion, corrugated and plank. Major stock types can be aliased using the Naming Convention Alias Manager.

Material

Material returns the material name of the stock.

Material Grade

Material Grade returns the material grade of the stock.

Nest Drawing

Nest Drawing returns the nest drawing name of plate nests.

Nominal Size
Nominal Size returns the nominal size of the pipe stock.

Part Side
Part Side returns the side of the ship on which the structure part lies.

Part Type
Part Type applies to Structure parts. Examples of part types are plate, stiffener, corrugation, faceplate, and plank. Part types can be aliases using the Naming Convention Alias Manager.

Planar Group Model Drawing
Planar Group Model Drawing returns the planar group model drawing that the structure part is modeled in.

Plate Stock Length
Plate Stock Length returns the length of the plate stock item that the structure part is made from.

Plate Stock Width
Plate Stock Width returns the width of the plate stock item that the structure part is made from.

Pressure Rating
Pressure Rating returns the pressure rating of the pipe stock.

Profile Plot Drawing
Profile Plot Drawing returns the profile plot drawing name of profile nests.

Project
Project returns the name of the ShipConstructor project.

Remnant
Remnant returns the name of the remnant that the plate nest was cut from.

Sheet Stock
Sheet Stock returns the name of the sheet stock that the HVAC stock is made from.

Stock
Stock returns the name of the stock that the part is made from.

System
System returns the spec level, system level, or branch level system name of the system of your entity. Which level is returned depends on your selection in the Tree Level drop-down list.

> Note: If the entity is in a system with a level higher than the selected system tree level, nothing is returned for that element

Thickness
Thickness returns the thickness of the plate stock that the plate part is made from.

Label Styles

Label styles are used in production output drawings. Label styles apply to annotations in production drawings.

Add a Label Style

To add a label style

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Production Output > Label Styles (page 339) to open the Label Styles window.
3. Click OK to close the Label Styles.

Dimension Styles

Dimension styles are used in production output drawings. The name of the dimension style must match the name in the dimension styles in the drawing.

Add a Dimension Style

To add a dimension style

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Production Output > Dimension Styles (page 339) to open the Dimension Styles window.
3. Click OK to close the Dimension Styles.

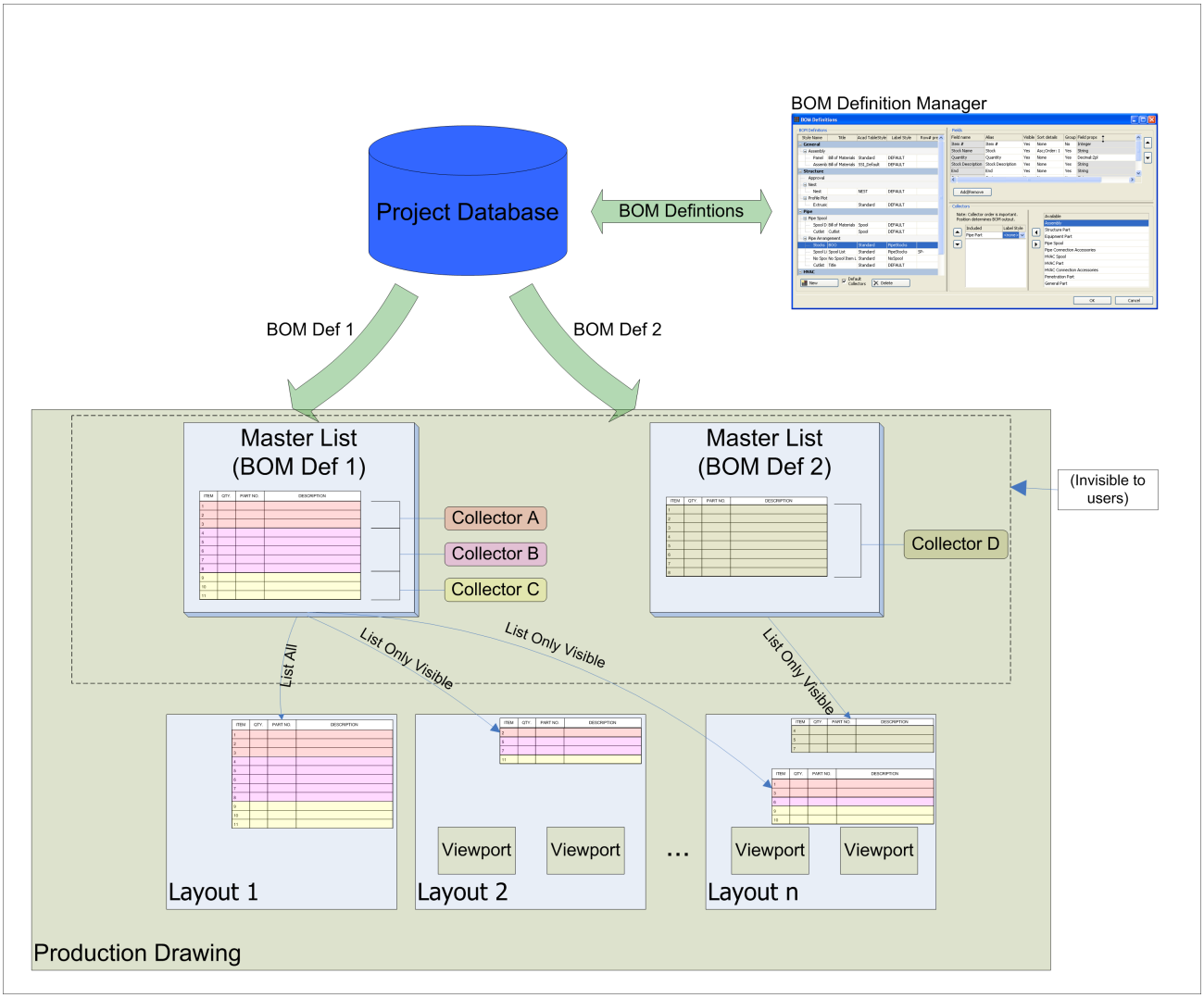
Bill of Materials

The ShipConstructor Bill of Materials (BOM) includes customizable and convenient functions:

- Management of all production drawing BOMs is performed through a unified interface, the BOM Definitions manager.
- BOMs have full merging, sorting, and grouping capability – even on hidden columns.
- You have control of object listing order in tables.
- Column heading aliasing.
- BOM tables use standard AutoCAD tables.
- BOM Functionality is identical regardless of drawing type.
- Consistent item numbering across all sheets.
- Updating without losing formatting.
- Revisions.

BOM Functionality Overview

ShipConstructor projects can have any number of user-customizable BOM definitions available for use in target production drawing types. As shown in the following diagram, multiple tables can be inserted in a production drawing, each referencing a master list.



One master list exists per BOM definition. The first time a BOM table is inserted in a drawing, its master list is created. Subsequent table insertions all reference the initial master list.

BOM content is determined at the time of BOM update. Collectors, specified by the BOM definition, organize the objects in the drawing into relevant groups (for example, Pipe, Accessories, and Spools) and process the BOM data into data blocks. The data blocks are then compiled into the master list with the tables being updated to reflect the master list data.

The order of the items listed in the BOM depends on primary and secondary factors. The primary order is determined by the collector. Each collector provides its own formatted data block. Within each data block, the row order is determined by the field sort order specified of the BOM definition.

Collectors process the data for their respective objects according to the format specified by the BOM definition. Therefore, each block of collector data is sorted according to the field sort order specified by the BOM definition.

Item numbering is performed on the master list at the time of BOM update. Tables inserted with the List Only Visible option retrieve their item numbering from the master list to ensure consistent numbering across all sheets. Item numbers can have user-defined prefix text (for example, SP–001).

To understand how collector use works, we will use an example: You have a Pipe arrangement drawing that contains structure, pipe, and HVAC, and you need two different bills of materials: a pipe stock BOM and spool list.

The pipe stock BOM must list all pipe items, HVAC items, and connection accessories, but not structure. This can be achieved by creating two BOM definitions, one for the stocks and one for the spool list.

The stock BOM should list the objects in a specific order: Pipe objects, HVAC objects, Pipe connection accessories, and then HVAC connection accessories.

To create a stock BOM that accomplishes the requirement, a BOM definition is created that includes the following collectors: Pipe Part, HVAC Part, Pipe Connection Accessories, and HVAC Connection Accessories. Order them as specified above; the BOM lists the items in the order specified

To create the spool list, another BOM definition is created that only includes the pipe spool collector.

Both of these BOMs can exist in the same drawing.

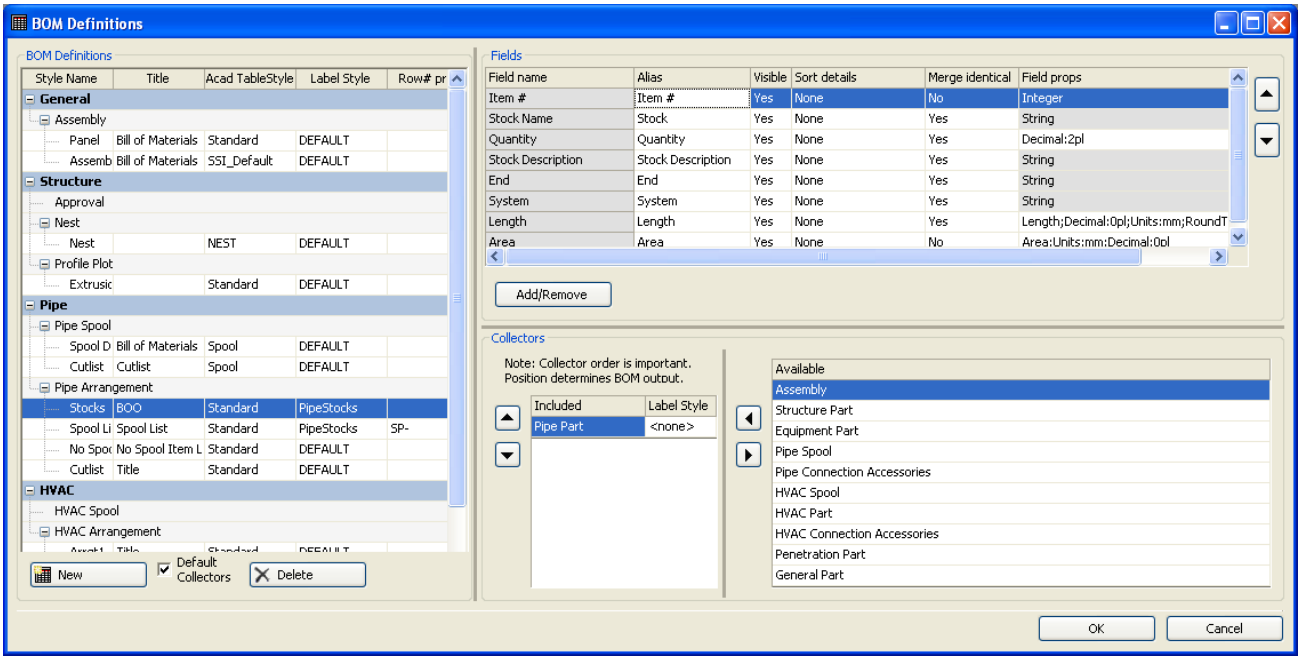
Virtually all other aspects of a BOM can be customized, including the following:

- Items included and their order.
- User-defined column header text.
- Each field can be sorted ascending or descending, in user-defined sequence.
- Rows can be merged dependent or independent of equivalent field data.
- Field properties such as units, display format (decimals or fractions), and rounding can be edited individually.

BOM Definitions Manager

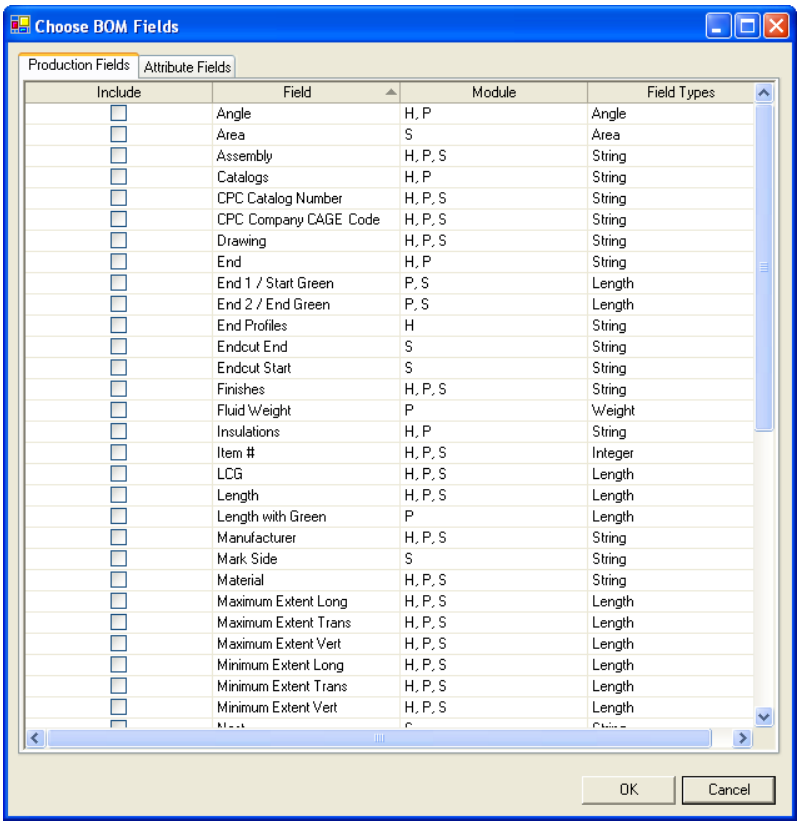
ShipConstructor integrates all Bill of Materials management through the BOM Definitions manager. The BOM Definitions manager lets you create and edit existing BOMs for all production drawing types.

Access the BOM Definitions manager through Manager (General > Production Output > Bill Of Materials (page 339)).



To create a BOM definition

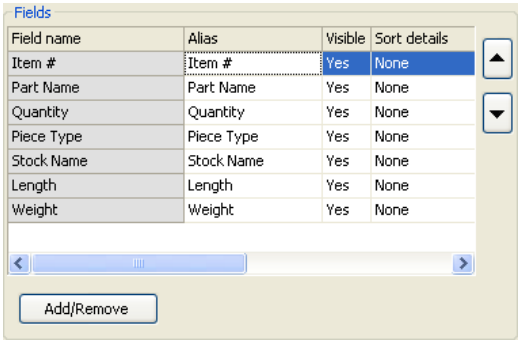
1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Production Output > Bill Of Materials (page 339) to open the BOM Definitions window.
3. Select the type of drawing in the BOM Definitions list that you want to create a BOM for.
4. Click New.
5. Enter the Style Name and fill in the remaining items in the BOM Definitions list.
6. With the style selected, Click Add/Remove in the Fields section.


Choose BOM Fields
Production Fields
Attribute Fields
Include
Field
Module
Field Types
Angle H, P Angle
Area S Area
Assembly H, P, S String
Catalogs H, P String
CPC Catalog Number H, P, S String
CPC Company CAGE Code H, P, S String
Drawing H, P, S String
End H, P String
End 1 / Start Green P, S Length
End 2 / End Green P, S Length
End Profiles H String
Endcut End S String
Endcut Start S String
Finishes H, P, S String
Fluid Weight P Weight
Insulations H, P String
Item # H, P, S Integer
LCG H, P, S Length
Length H, P, S Length
Length with Green P Length
Manufacturer H, P, S String
Mark Side S String
Material H, P, S String
Maximum Extent Long H, P, S Length
Maximum Extent Trans H, P, S Length
Maximum Extent Vert H, P, S Length
Minimum Extent Long H, P, S Length
Minimum Extent Trans H, P, S Length
Minimum Extent Vert H, P, S Length
OK
Cancel

7. Place a check next to each field you want to include in the BOM. For BOMs that include structure parts only include fields that have Module S. P is for Pipe, H is for HVAC.

Note: For structure BOMs you should always include Part Name and Quantity fields. Item # is required for labeling parts.

8. Click OK to return to the BOM Definitions window. You will see that the fields are added to the fields list.

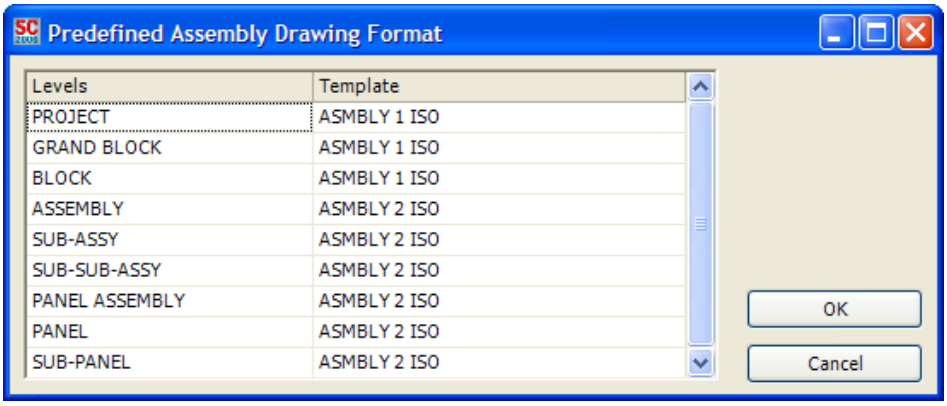


9. Use the and buttons to organize the order of the columns. The top field will be the leftmost column in the table.
10. Review the Collectors sections to ensure the types of parts that you want in the BOM table are in the Included collector list. Assign a label style to each collector that you want labeled.
11. Click OK to close the BOM Definitions.

Predefined Assembly Format

The predefined assembly format allows you to set up which templates are used for each assembly level when the Use Predefined Templates option is selected.

Setting up the predefined templates requires an Assembly Template drawing. See Assembly Drawings (page 218).



User-Defined Attributes

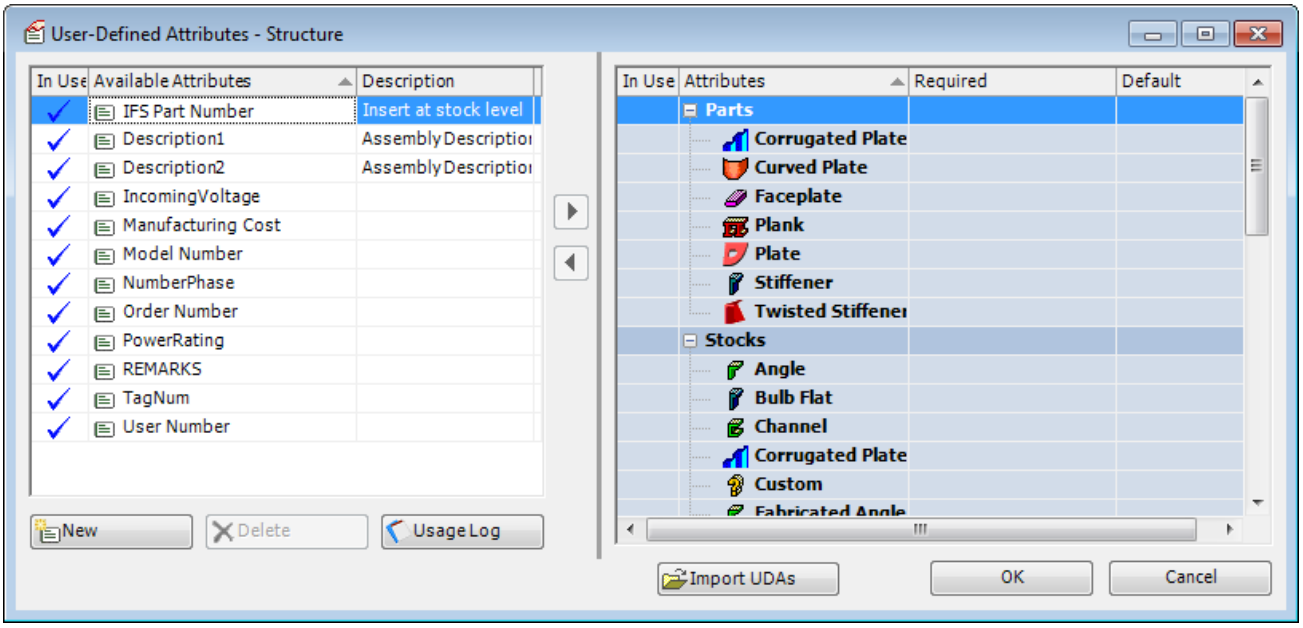
User-defined attributes let you define custom attributes to hold arbitrary data associated with a part (for example, model number, process codes, or custom identifiers), making it easier to manage parts. You can set up user-defined attributes for Structure, Pipe, and HVAC. Attributes assigned to parts can have four states: not required, required and deferrable, required and not deferrable, and finally, required default deferred. Stock attributes have two: required or not required.

The same attribute can be assigned to different object types, each with their own required state and default value.

Create a User-Defined Attribute

To create a user-defined attribute

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > User-Defined Attributes (page 345) to open the User-Defined Attributes window.



3. Click New.
4. Enter a name for the attribute and press Enter.
5. Optionally, add a description.
6. Click OK to close the User-Defined Attributes window.

Assign a User-Defined Attribute to a Part or Stock

To assign a user-defined attribute to a part or stock

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > User-Defined Attributes (page 345) to open the User-Defined Attributes window.
3. Select the attribute on the left side of the window
4. Select the part or stock on the right side of the window.
5. Click the right arrow button .

   The attribute is now listed beneath the part or stock.

   Note: To remove an attribute from a part or stock, select the attribute on the right side of the window and click the left arrow button .
6. Set the required state.
7. Optionally, set a default value.
8. Click OK to close the User-Defined Attributes window.

   When you add a user-defined attribute to stocks or a stock size, the attribute will appear as an additional column in the stock list (or stock size list). Required attributes appear with a * after their name.

   When you add a user-defined attribute to parts, the attribute will appear on the User Attributes tab of the Part Properties window. Required attributes will already be assigned to the part. See Part User-Defined Attributes (page 199).

Default Values on Required User-Defined Attributes

Required user attributes do not have to have a default value, with one notable exception. If the attribute is either “required” (for stocks and stock sizes) or “required not deferrable” (for parts), and those objects already exist, then a default value must be provided.

For example, if an attribute is assigned to the plate part type, and it is set as “required not deferrable”, and there are no plate parts created yet, then the attribute does not need a default value. However, if some plate parts already exist, a default value must be provided.

This is because an object cannot have a required attribute without a value: all the existing objects will need a value for the new attribute.

This requirement prevents the User-Defined Attributes window from closing, if there are any missing default values. A message window will appear listing the attribute name and the object type for that attributes.

Import a User-Defined Attribute

To export a user-defined attribute

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > User-Defined Attributes (page 345) to open the User-Defined Attributes window.
3. Click Import UDAs button
4. Browse to the selected xml or pro file
5. Click OK

Stock Catalog

Stocks are the raw materials that shipyard workers fabricate parts from. There are several different types of structure stocks:

Plate stocks – These are solid plates used to fabricate plate parts. See Plate Parts (page 158).

Corrugated plate stocks – These are corrugated plates used to fabricate certain plate parts. See Corrugated Plates (page 180).

Profile stocks – These are used to fabricate stiffeners and faceplates. See Stiffeners (page 167) and Faceplates (page 178).

Plank stocks – These stocks are used to automatically divide a predefined area (for example, a deck or bulkhead) into planks. See Planks (page 189).

Create a Stock

To create a stock

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Stock Catalog (page 342) to open the Structure Stock Catalog.
3. Select the type of stock you want to create.

   To create a plate stock, corrugated plate stock, or plank stock, select Plate, Corrugated Plate, or Plank.

   To create a profile stock, select the type of profile stock (for example, Angle or Bulb Flat).
4. Click New.
5. Type a name for the stock and press Enter.
6. Set the Stock Properties (see below). See Structure Stock Catalog (page 343) for more details.
7. Click Edit Sizes to open the Structure Stock Available Sizes window and specify the available sizes.
8. Click Apply to save the changes.

9. Click OK to close the Structure Stock Catalog.

Plate Stock Properties

Name – The name of the stock.

Thickness -- The thickness of the plate stock.

Material – The material the stock is made of.

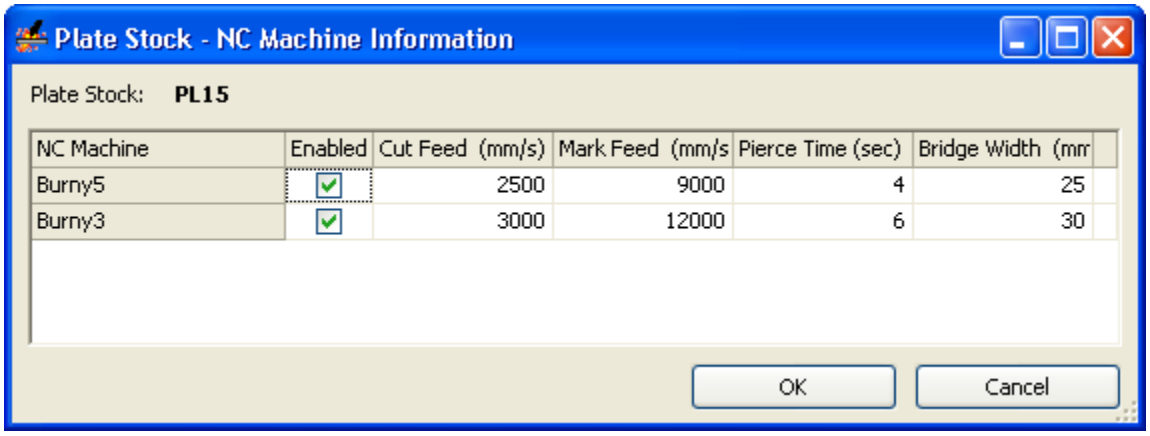
Color – The color used to draw parts made of this stock

Flg Inner Rad(X Thick) – This value is multiplied by the thickness of the plate and used when flanging a plate. The value represents the inside radius of the flange. It affects both the solid representation of the flange and the expanded representation. Each flange instance can override this value.

Nest Part Offset – The minimum distance between two nested parts on this plate stock.

Nest Edge Offset – The minimum distance between the edge of a nested part and the edge of the plate stock.

NC Machine – Lets you set properties for the plate stock that relate to the NC machine that will be used to cut parts nested on this plate stock.



NC Machine, Enabled – This is the NC machine that will cut any nests created from this stock. The Enabled check box must be checked to enter the following information. A check in the check box indicates that the displayed plate stock can be cut on that NC machine.

Cut Feed – This is the rate at which the NC machine will be able to cut this plate stock. ShipConstructor uses this value to estimate the time necessary to process the nest.

Mark Feed – This is the rate at which the NC machine will mark this plate stock. ShipConstructor uses this value to estimate the time necessary to process the nest.

Pierce Time – This is the amount of time required for the NC machine to pierce the plate stock. ShipConstructor uses this value to estimate the time needed to process the nest.

Bridge Width – This is the width of bridges between parts nested on this plate stock. (Bridges connect two adjacent plate parts.) See Set Up Bridges (page 282).

Manufacturer – The manufacturer of the stock (optional).

Description – A description of the stock (optional).

Max Strain Threshold (%) – The maximum allowable strain on a plate when expanding a curved plate. A warning appears when the threshold is exceeded. Positive strain values refer to expansion. Negative strain values refer to compression. This value is determined by experience for different plate materials and thicknesses using different forming methods.

Min Strain Threshold (%) - The minimum allowable strain on a plate when expanding a curved plate. A warning appears when the threshold is exceeded. Positive strain values refer to expansion. Negative strain values refer to compression. This value is determined by experience for different plate materials and thicknesses using different forming methods.

Corrugated Plate Stock Properties

For a description of general stock properties, see Plate Stock Properties (page 52). In addition to the properties, you can edit a corrugated plate stock's shape drawing.

Shape Drawing

Click to create or edit the corrugated plate stock's shape drawing (using polylines and points). The shape must be an open 2D polyline in the WCS XY plane. This polyline, and the bend line point markers, must be the only objects in the drawing. The lower left corner of the polyline is the insertion point when you create the solid. Create points using AutoCAD's **POINT** command, and place them at key locations on the profile to represent bending points. These will be used to generate bend lines when generating expanded geometry. Bend lines are straight lines on the surface of the flat plate used to form the corrugation profile. They are always parallel to each other.

Corrugation profile with highlighted bend line markers

Profile Stock Properties

For a description of general stock properties, see Plate Stock Properties (page 52). In addition to the general properties, profile stock properties include the following.

Shape – The shape of the stock's cross section. To create or edit stock profile shapes, see Create a Profile Stock Shape (page 54).

Kerf – The gap, or kerf, between parts in a profile nest.

Smallest Remnant – The minimum length of this stock that can be used for creating stock remnants. This is used during profile nesting. Left over pieces smaller than this value will be considered scrap.

Cutout – Lists the cutout type.

> Note: The Cutout option does not appear if you have not defined any cutout types. To define cutout types, click Cutout Types in the Structure Stock Catalog.

Plank Stock Properties

For a description of general stock properties, see Plate Stock Properties (page 52). In addition to the general properties, plank stocks have the following properties and options.

Width – The effective width of an individual plank (that is, the total plank width minus any overlap).

Shape Drawing – Click to create or edit the plank stock's cross section shape. The shape can consist of circles or polylines. The shape must represent one closed region but can have holes.

Edit a Stock

To edit a stock

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Stock Catalog (page 342) to open the Structure Stock Catalog.
3. Select the stock type to list stocks of that type.
4. Set the Stock Properties. See Create a Stock (page 51).
5. Click Apply to save the changes.

6. Click OK to save the changes and close the Structure Stock Catalog.

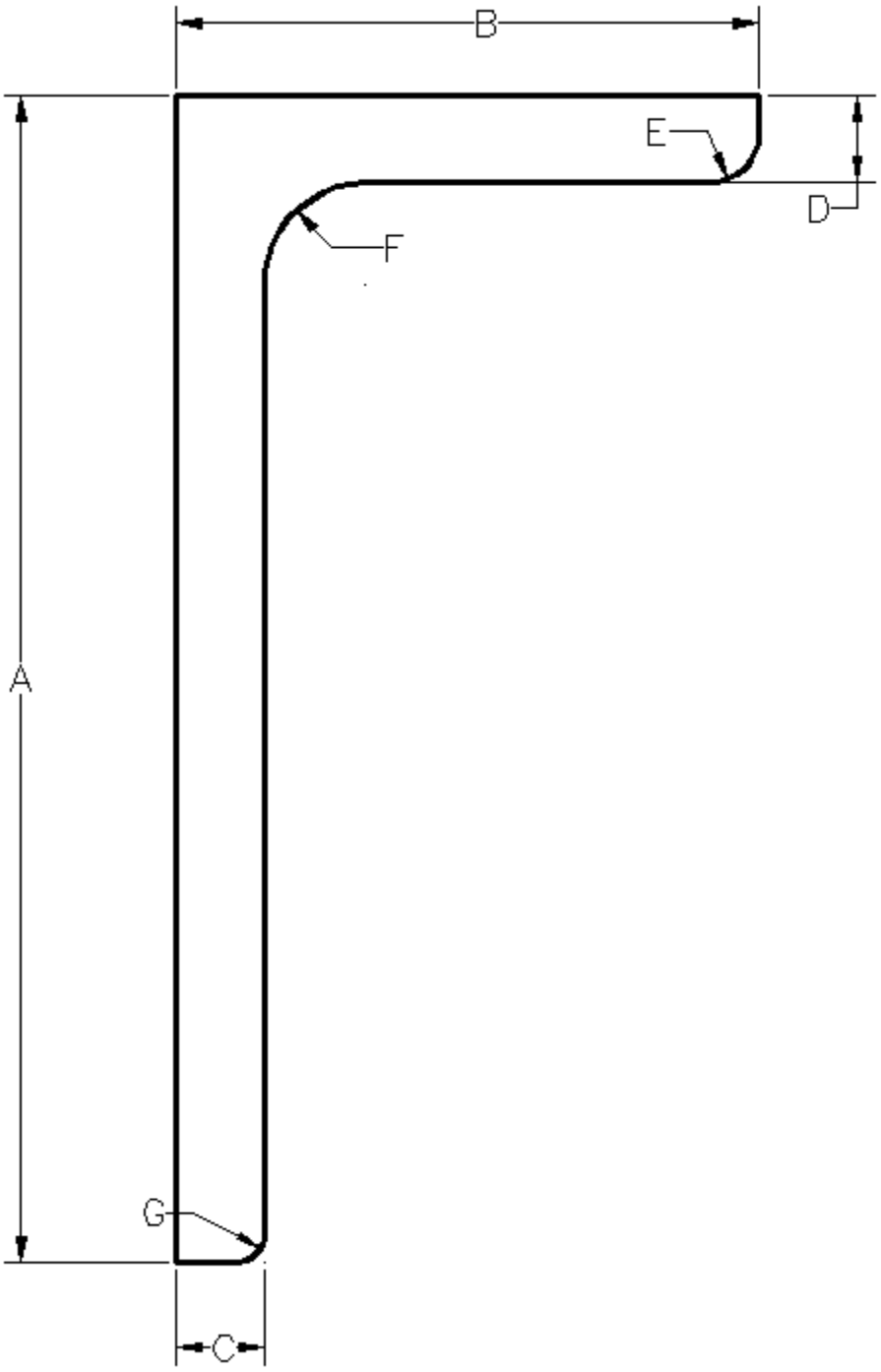
Create a Profile Stock Shape

To create a profile stock shape

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Stock Catalog (page 342) to open the Structure Stock Catalog.
3. Click Edit Shapes to open the Profile Shapes window.
4. Under Profile Standards select the type of profile stock that you want to create a profile shape for.
5. Click New.
6. Type a name for the profile shape and press Enter.
7. Set the profile stock Properties (see below).
8. Click OK to close the Profile Shapes window.
9. Click OK to close the Structure Stock Catalog.

Angle Stock Shape Properties

Shape Name – The name of the shape. The shape is usually named after the profile stock.



Web Height – A

Flange Length – B

Web Thickness – C

Flange Thickness – D

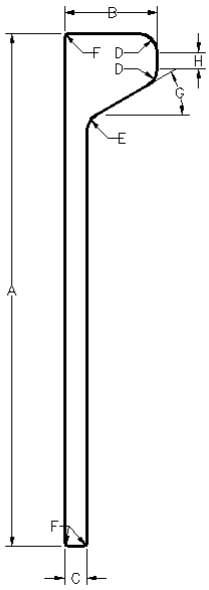
Flange Fillet Radius – E

Corner Fillet Radius – F

Web Fillet Radius – G

Bulb Flat Stock Shape Properties

Shape Name -- The name of the shape. The shape is usually named after the profile stock.



Web Height – A

Flange Length – B

Web Thickness – C

Bulb Radius – D

Inside Corner Radius – E

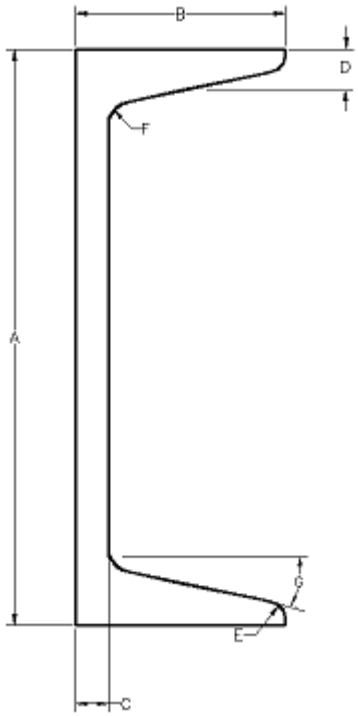
Web Fillet Radius – F

Flange Taper Angle – G

Bulb Nose Height – H

Channel Stock Shape Properties

Shape Name -- The name of the shape. The shape is usually named after the profile stock.



Web Height – A

Flange Length – B

Web Thickness – C

Flange Thickness – D

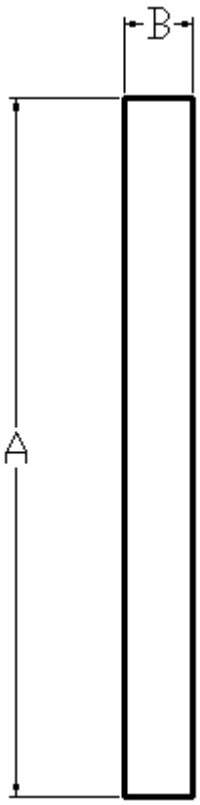
Flange Fillet Radius – E

Corner Fillet Radius – F

Flange Taper Angle – G

Flat Bar Stock Shape Properties

Shape Name -- The name of the shape. The shape is usually named after the profile stock.

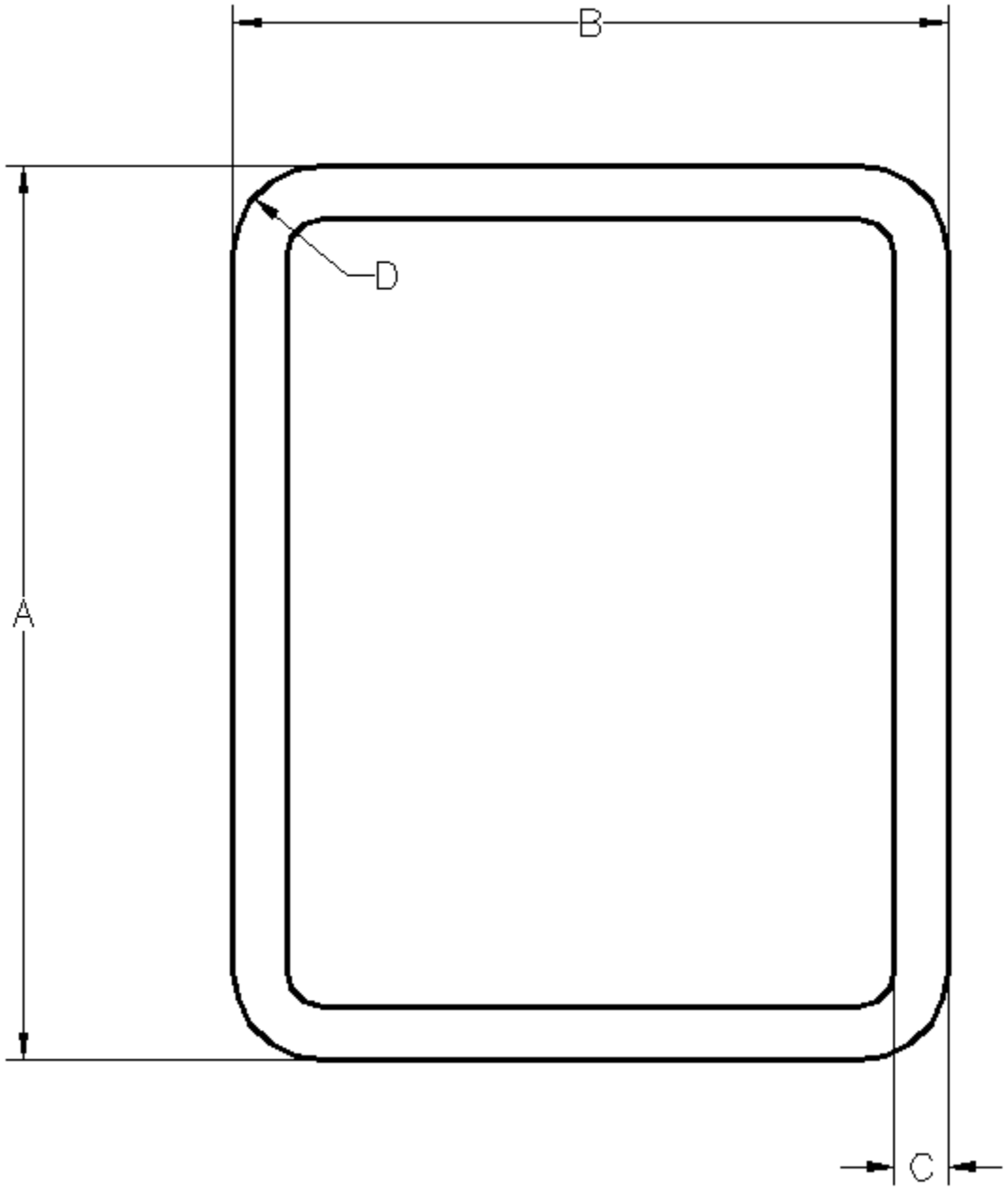


Width – A

Thickness – B

Rect Tube Stock Properties

Shape Name -- The name of the shape. The shape is usually named after the profile stock.



Height – A

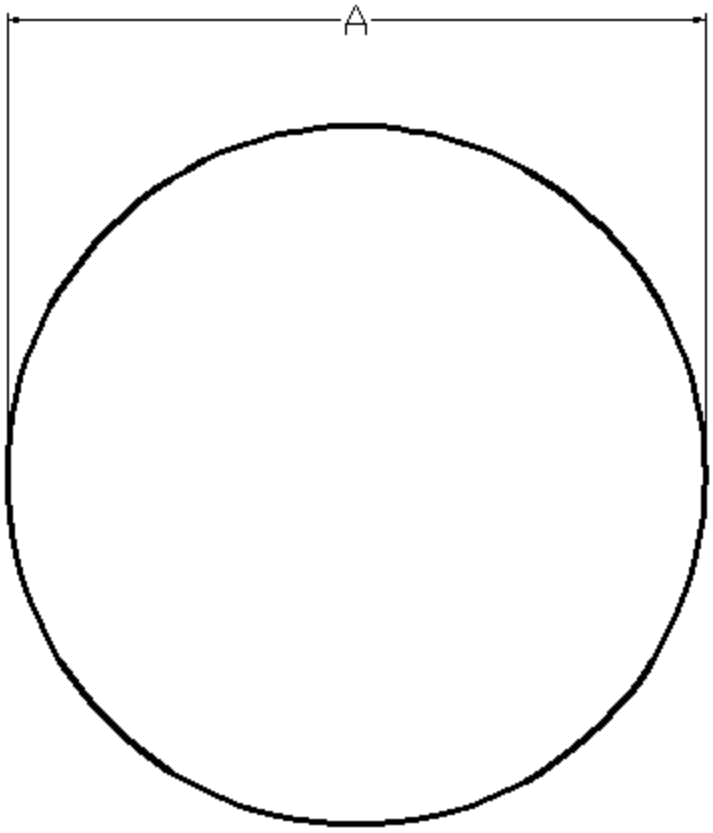
Width – B

Thickness – C

Corner Radius – D

Round Bar Stock Shape Properties

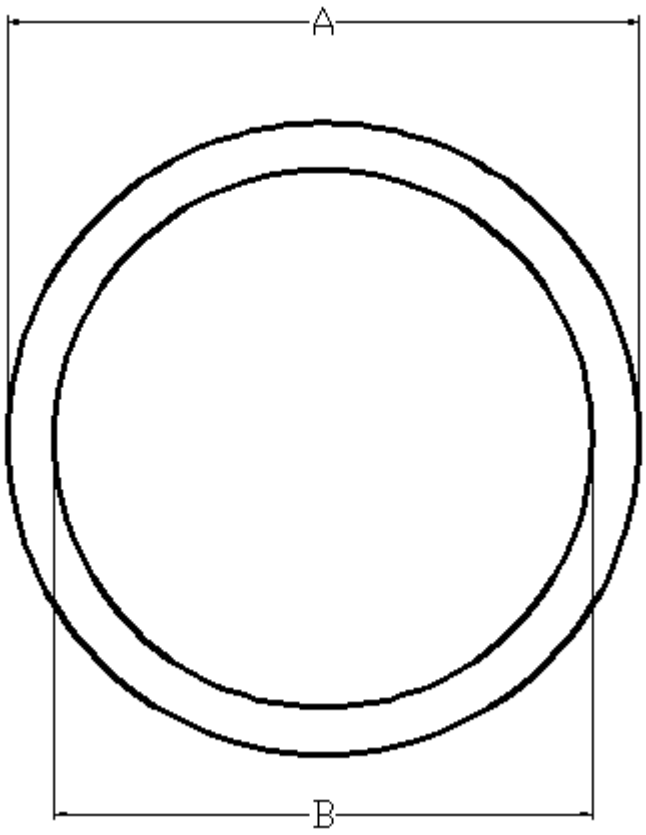
Shape Name -- The name of the shape. The shape is usually named after the profile stock.



Outer Diameter – A

Structural Pipe Stock Shape Properties

Shape Name -- The name of the shape. The shape is usually named after the profile stock.



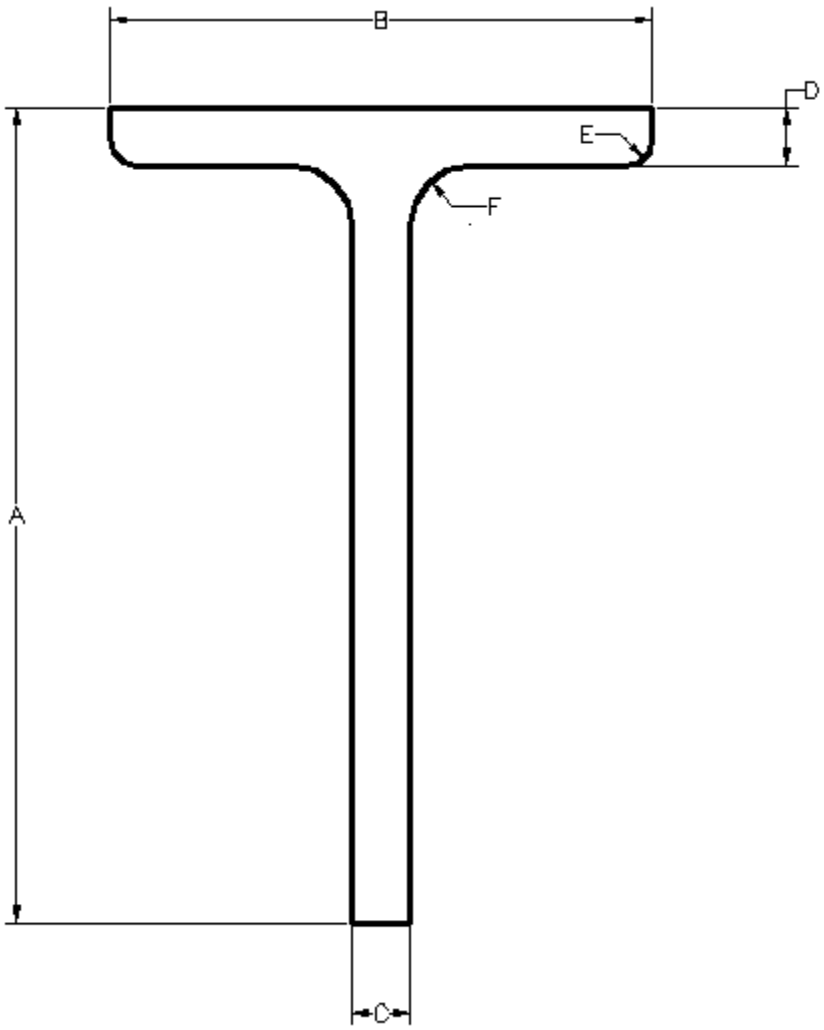
Outer Diameter – A

Inner Diameter – B

Tee Stock Shape Properties

Shape Name -- The name of the shape. The shape is usually named after the profile stock.

Specialized Type -- None



Web Height – A

Flange Length – B

Web Thickness – C

Flange Thickness – D

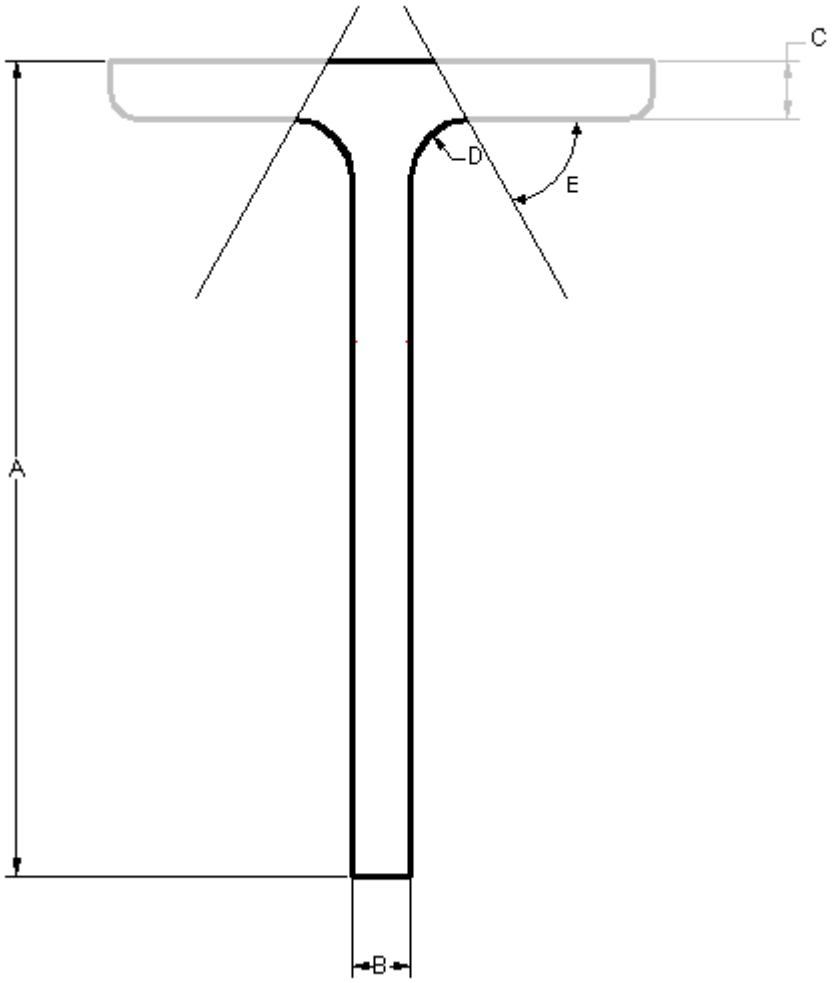
Flange Fillet Radius – E

Corner Fillet Radius – F

Tee Stock – IJC Shape Properties

Shape Name -- The name of the shape. The shape is usually named after the profile stock.

Specialized Type -- IJC



Web Height – A

Web Thickness – B

Flange Thickness – C

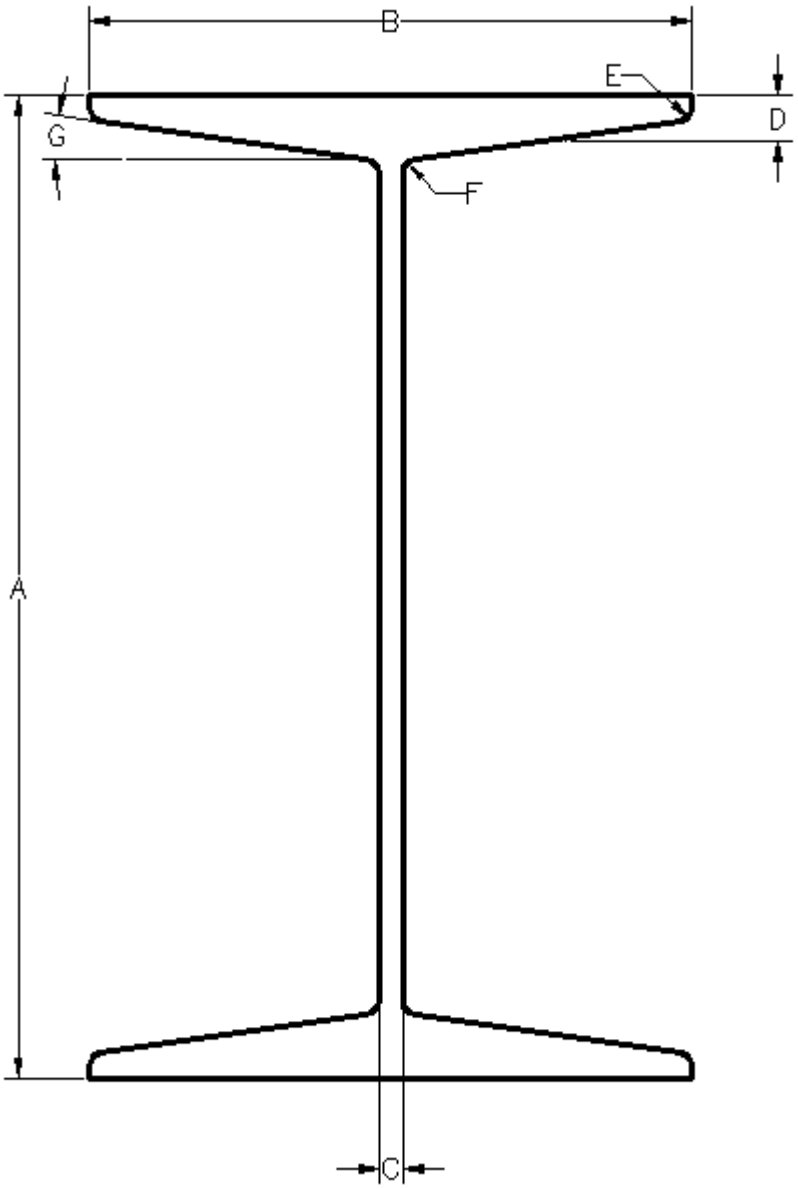
Corner Fillet Radius – D

Flange Bevel Angle – E

W Stock Shape Properties

Shape Name -- The name of the shape. The shape is usually named after the profile stock.

Specialized Type -- None



Web Height – A

Flange Length – B

Web Thickness – C

Flange Thickness – D

Flange Fillet Radius – E

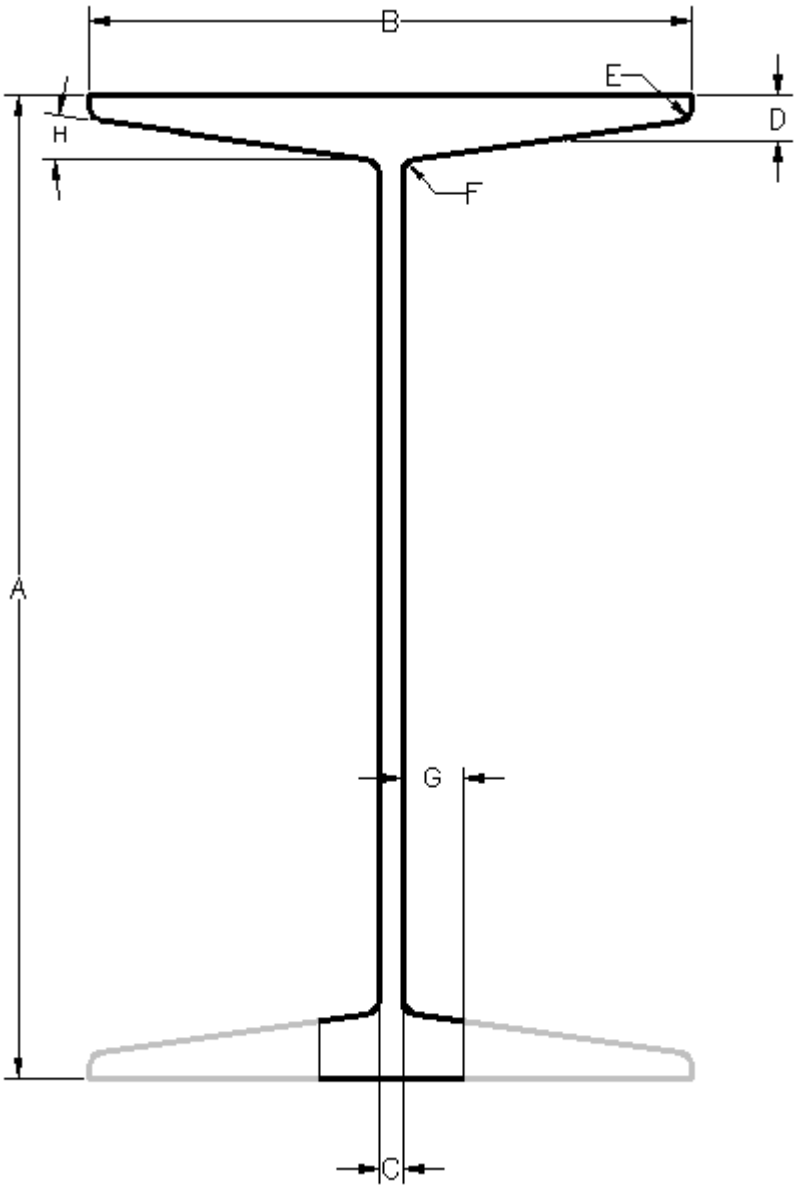
Corner Fillet Radius – F

Flange Taper Angle – G

W Stock – W-T Shape Properties

Shape Name -- The name of the shape. The shape is usually named after the profile stock.

Specialized Type – W-T



Web Height – A

Flange Length – B

Web Thickness – C

Flange Thickness – D

Flange Fillet Radius – E

Corner Fillet Radius – F

Bottom Flange Offset – G

Flange Taper Angle – H

Custom Stock Shape Properties

Shape Name -- The name of the shape. The shape is usually named after the profile stock.

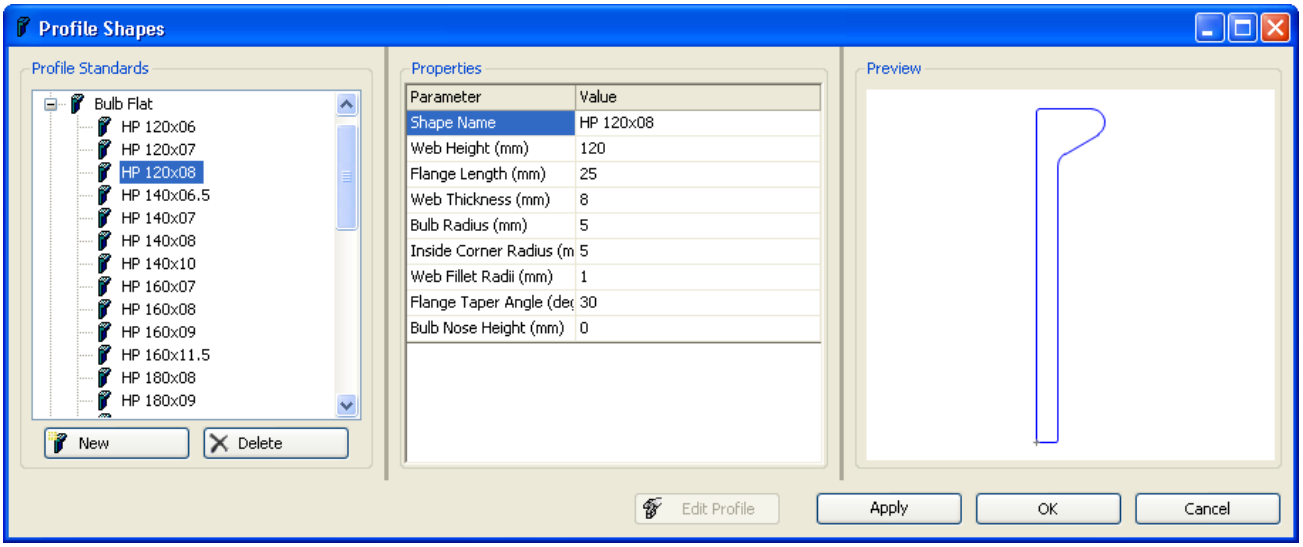
The shape can be only one contiguous region that is defined by multiple closed polylines or circles. The moldline point of the shape is taken to be the (0,0) point in the cross section drawing.

Note: Component trimming of stiffeners does not apply to custom shaped stocks.

Edit a Profile Stock Shape

To edit a profile stock shape

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Stock Catalog (page 342) to open the Structure Stock Catalog.
3. Click Edit Shapes to open the Profile Shapes window.



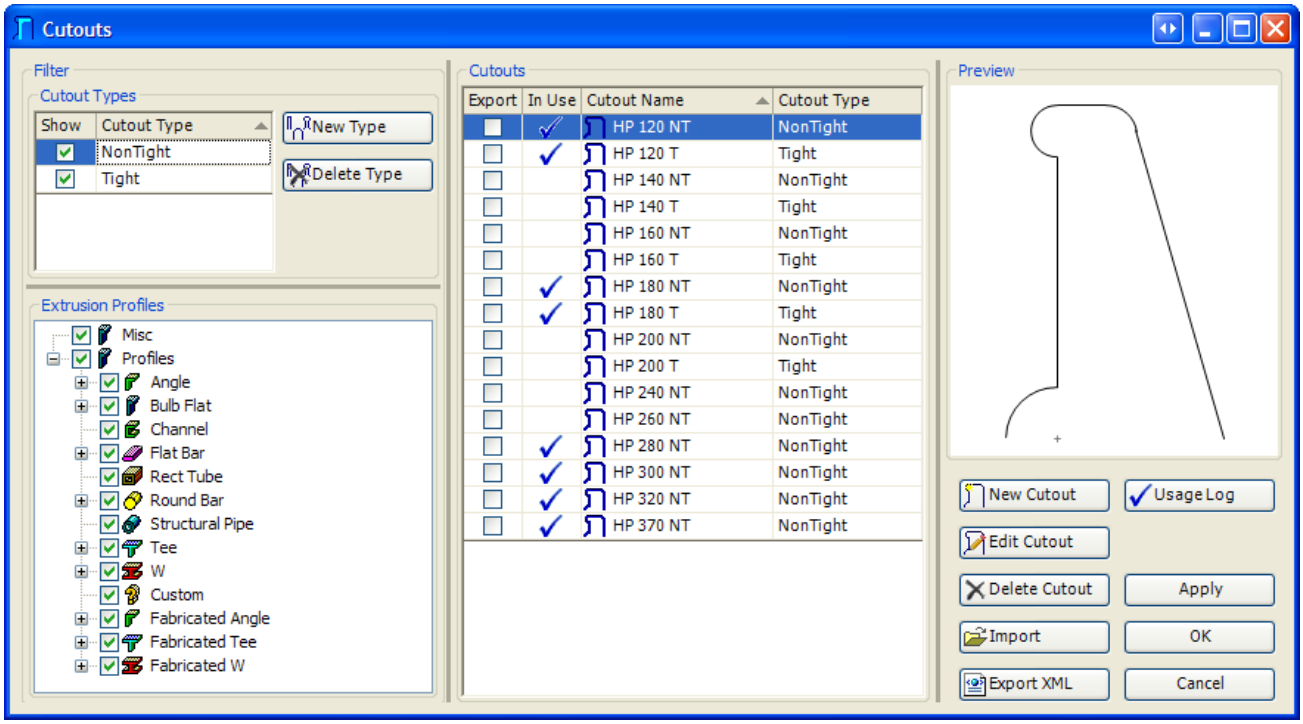
4. Under Profile Standards select the profile shape.
5. Set the profile stock Properties (see Create a Profile Stock Shape (page 54)).
6. Click OK to close the Profile Shapes window.
7. Click OK to close the Structure Stock Catalog.

Create a Cutout Type

Cutouts are added to the plate part after the part is defined. The part solid is automatically updated with the production information when a cutout is added. Before a profile can create a cutout through a plate part, you must create a cutout type. That is, the stock of the profile must have a cutout shape associated with the stock.

To create a cutout type

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Stock Catalog (page 342) to open the Structure Stock Catalog.
3. Click Cutout Shapes to open the Cutouts window.



4. Click New Type.
5. Type a name for the cutout type and press Enter.
6. Click OK to save the changes.

The new cutout type will be listed as a column in the stock list in the Structure Stock Catalog.

Create a Cutout

To create a cutout shape and associate it with a stock

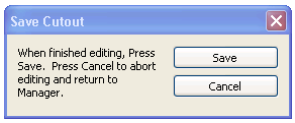
1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Stock Catalog (page 342) to open the Structure Stock Catalog.
3. Click Cutout Shapes to open the Cutouts window.
4. Click New Cutout. Selecting a Cutout Type and a Profile will automatically name the new cutout from the Profile with capital letters from the Cutout Type. Typically, you will name the cutout from the size of the profile.
5. Type a name for the cutout and press Enter.
6. Click Edit Cutout to edit the shape of the cutout. You cannot finish creating a cutout without editing it. You will see a when no polyline is assigned to a cutout.
7. ShipConstructor will switch applications to AutoCAD.
8. If you have a profile selected or the cutout is associated with a stock, you will see the following window.

9. Click Yes to insert a polyline for the cross section of the profile stock. This polyline can remain after you are finished because it is flagged as being a profile and not the cutout.

10. You can now draw the cutout shape. If you are creating a cutout for a profile, be aware of the position and orientation of the profile.

    Note: The cutout does not need to extend below Y=0 because ShipConstructor extends the ends of the polyline to account for cases when the cutout would not intersect the edge of a plate. The cutout should be an open polyline.

11. You will see a window that lets you save the cutout to the database or cancel the edit.



12. Click Save to save the cutout to the database.
13. Select the correct cutout type.
14. Click OK.
15. You will see the Cutout Type columns on the right side of the Stock Properties list.
16. Select the profile stock to add the cutout to.
17. In the Cutout Type column, select the correct cutout shape from the drop-down list.

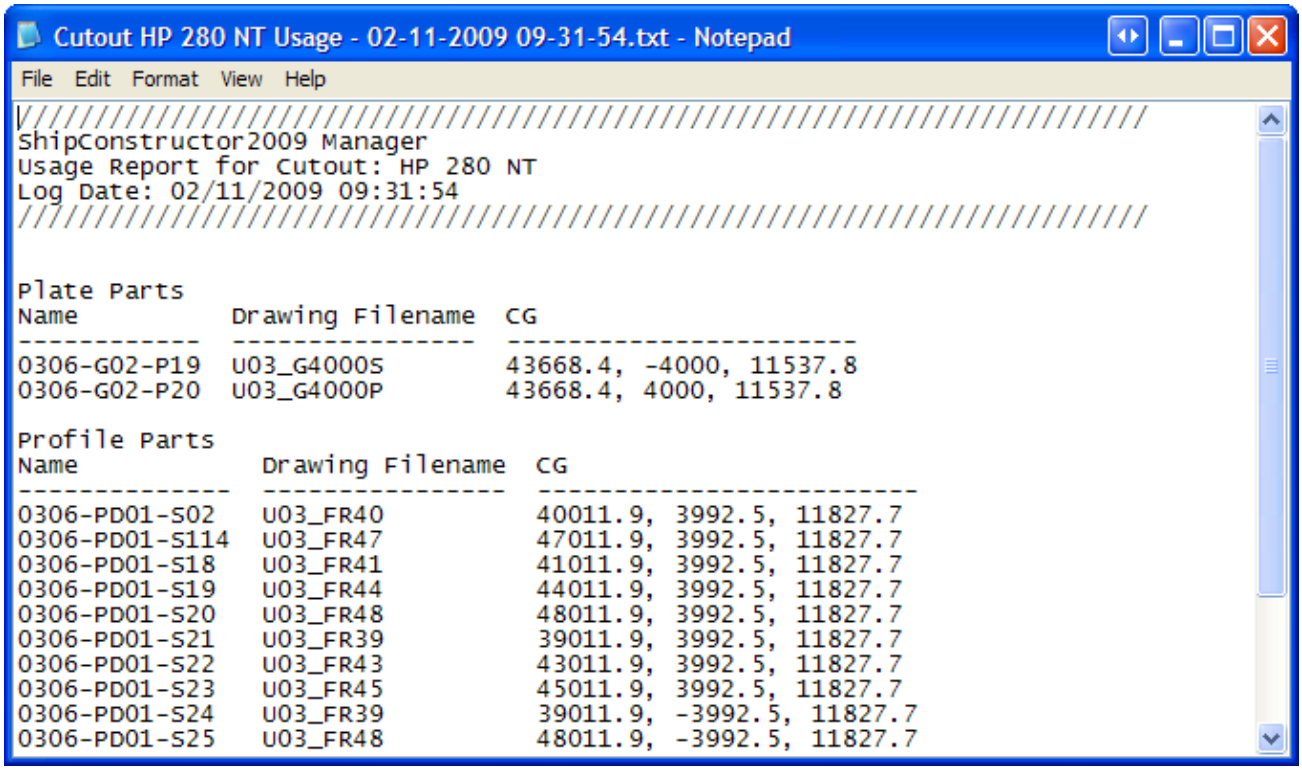
View Cutout Usage

It is possible to see a list of all the parts that use a selected cutout. A check mark in the In Use column indicates that the cutout is being used by parts.

To list all parts using a specific cutout

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Stock Catalog (page 342) to open the Structure Stock Catalog.
3. Click Cutout Shapes to open the Cutouts window.
4. Select the cutout you want.
5. Click Usage Log.

   ShipConstructor displays a list of all parts using the selected cutout.



Import Stocks

To import stocks

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Stock Catalog (page 342) to open the Structure Stock Catalog.
3. Click Import to open a File Browser.
4. Select a project file (*.PRO) or XML file (*.XML).
5. Click Open.

Export Stocks

Exporting a stock includes the sizes of the stock, endcuts associated with the stock, and cutouts associated with the stock.

To export stocks

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Stock Catalog (page 342) to open the Structure Stock Catalog.
3. In the structure stock list, check the check boxes for the stocks that you want to export.
4. Click Export XML to open a File Browser.
5. Type a name for the XML file.
6. Click Save.

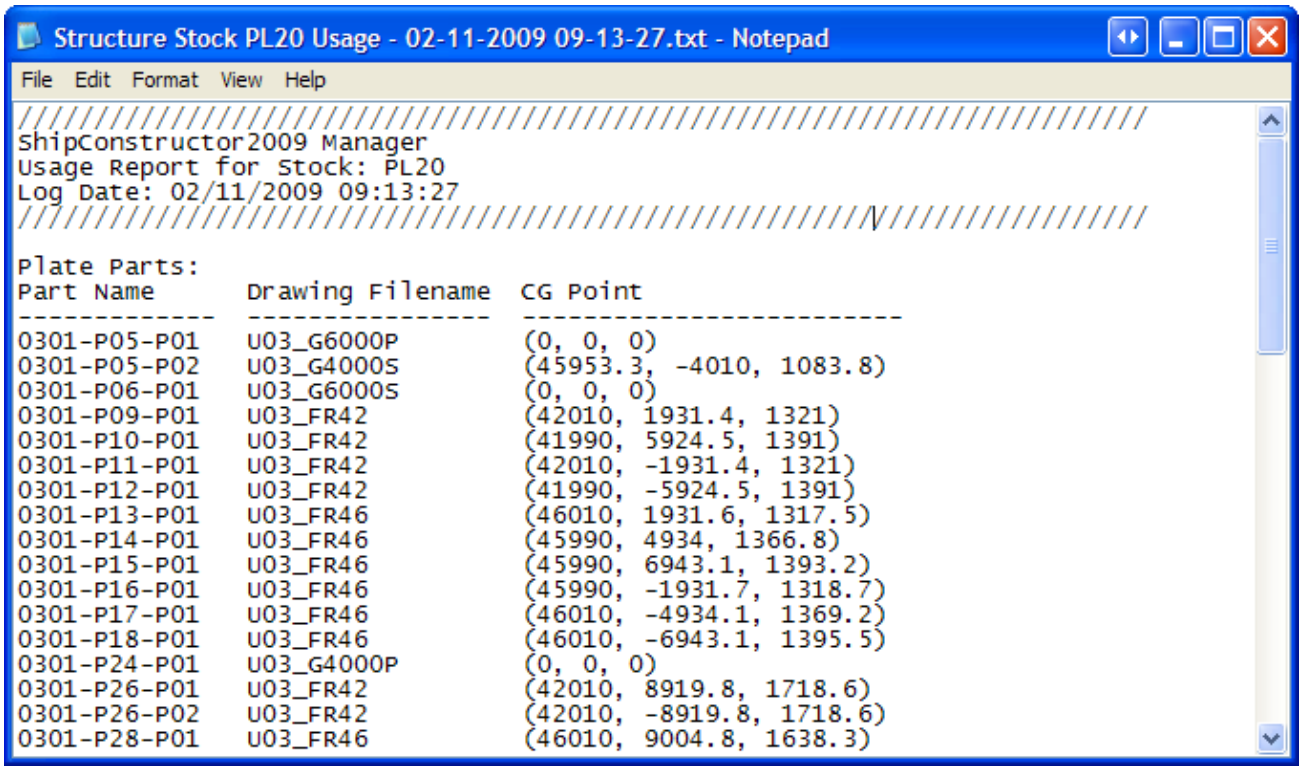
View Stock Usage

It is possible to see a list of all the parts that use a selected stock. A check mark in the In Use column indicates that the stock is being used by parts.

To list all parts using a specific stock

6. Choose ShipConstructor > Manager (page 383) to open Manager.
7. Choose Structure > Stock Catalog (page 342) to open the Structure Stock Catalog.
8. Select the stock you want.
9. Click Usage Log.

   ShipConstructor displays a list of all parts using the selected stock.



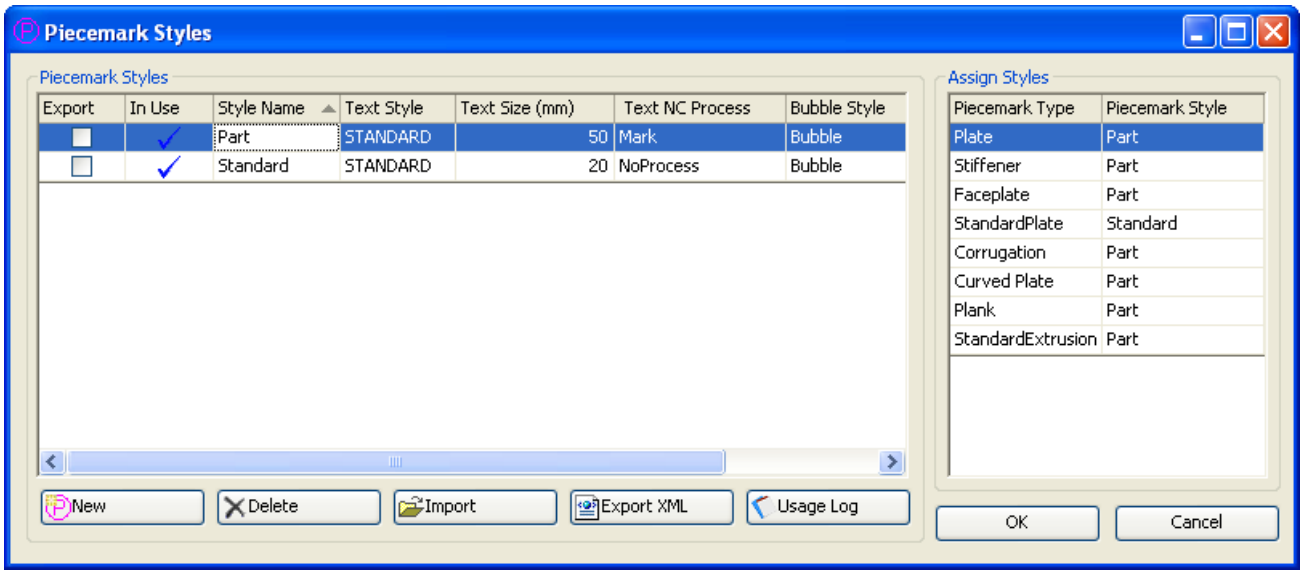
Piecemark Styles

Piecemark styles control how piecemarks are formatted on plates, stiffeners, faceplates, and standard parts.

Create a Piecemark Style

To create a piecemark style

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Piecemark Styles (page 345) to open the Piecemark Styles window.



3. Click New.
4. Enter a Style Name for the piecemark style and press Enter.
5. Set the options for the piecemark style:

   Text Style – The text style for the piecemark. See Text Styles (page 35).

   Text Size – The size of piecemark text.

   Text NC Process – Whether the piecemark's text will be marked on parts (Mark) or not (No Process) during NC cutting. This option only applies to parts that are NC cut.

   Bubble Style – The style of bubble surrounding the piecemark's text.

   Bubble NC Process – Whether the piecemark will be marked on parts (Mark) or not (No Process) during NC cutting.
6. In the Assign Styles list, change the Piecemark Style for the Piecemark Type you want to assign the new style to. Click in the area of the specific Piecemark Style to show the arrow for the drop-down list.
7. Click OK to close the Piecemark Styles window.

Edit Piecemark Style

To edit a piecemark style

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Piecemark Styles (page 345) to open the Piecemark Styles window.
3. Double-click the field to edit and enter a new value.
4. Click OK to close the Piecemark Styles window.

Activate a Piecemark Style

To activate a piecemark style

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Piecemark Styles (page 345) to open the Piecemark Styles window.
3. In the Assign Styles pane, select the Piecemark Type.
4. Select the Piecemark Style for the selected Piecemark Type.
5. Click OK to close the Piecemark Styles window.

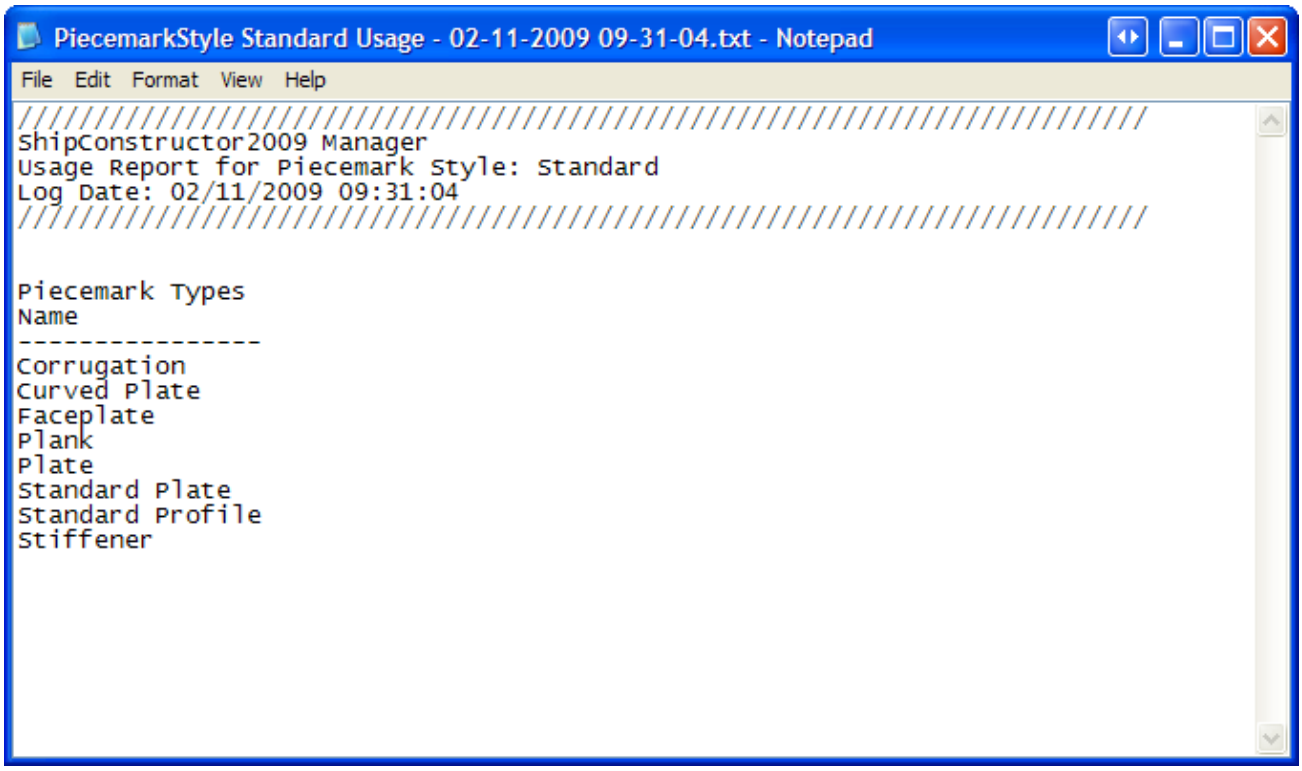
View Piecemark Usage

It is possible to see a list of all the part types that use a selected piecemark style. A check mark in the In Use column indicates that the piecemark style is being used by part types.

To list all part types using a specific piecemark style

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Piecemark Styles (page 345) to open the Piecemark Styles window.
3. Select the piecemark style you want.
4. Click Usage Log.

   ShipConstructor displays a list of all part types using the selected piecemark style.



Orientation Icon

An orientation icon is a symbol that you can insert onto plate parts to identify their orientation. Orientation icons can simplify assembly and help avoid costly mistakes.

An orientation icon consists of two direction arrows and keywords indicating the directions. There are two sets of direction keywords, Starboard/Port Only and Inboard/Outboard Only. You can choose to use either set or both sets. You can also specify the text used for each direction.

To set up an orientation icon

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Orientation Icon (page 346) to open the Orientation Icon window.
3. To insert an orientation icon onto plate parts by default, turn on Insert Orientation Icons in Plate Parts.
4. Set the Properties.
5. To specify the text used for direction keywords (in any language), set the Direction Aliases.
6. Click OK to close the Orientation Icon window.

When creating parts, you can uncheck the Insert Orientation Icon checkbox to not insert an orientation for that part. This setting is remembered when creating subsequent plate parts in that drawing.

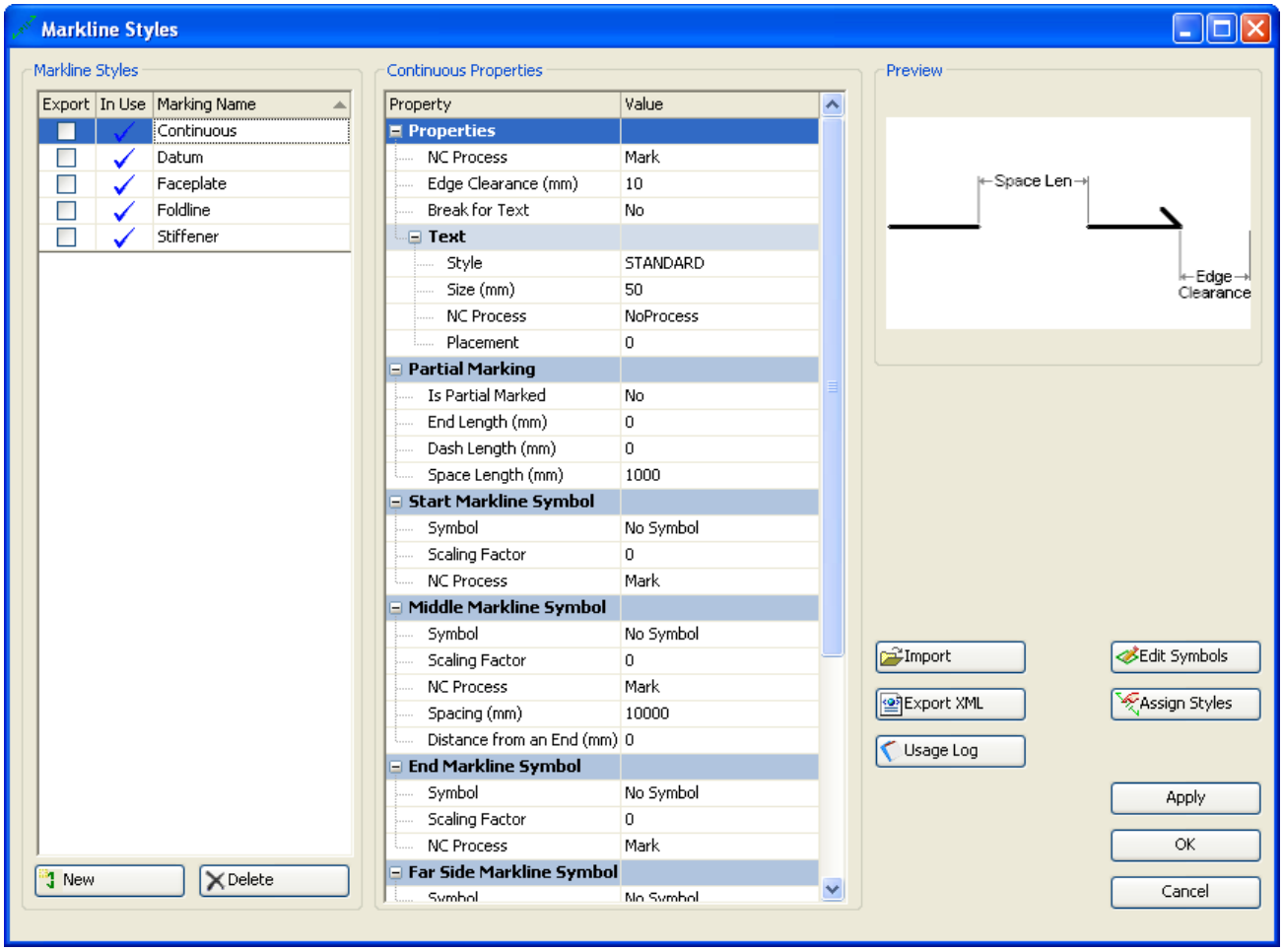
Markline Styles

Markline styles control how marklines are formatted on stiffeners, flange foldlines, corrugated plate foldlines, and datum lines. All marklines on plate parts have a style that is controlled in the Markline Styles window.

Create a Markline Style

To create a markline style

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Markline Styles (page 348) to open the Markline Styles window.



3. Click New.
4. Enter a name for the markline style and press Enter.
5. Set the markline style Properties. See Structure > Plates > Markline Styles (page 348).
6. Click OK to close the Markline Styles window.

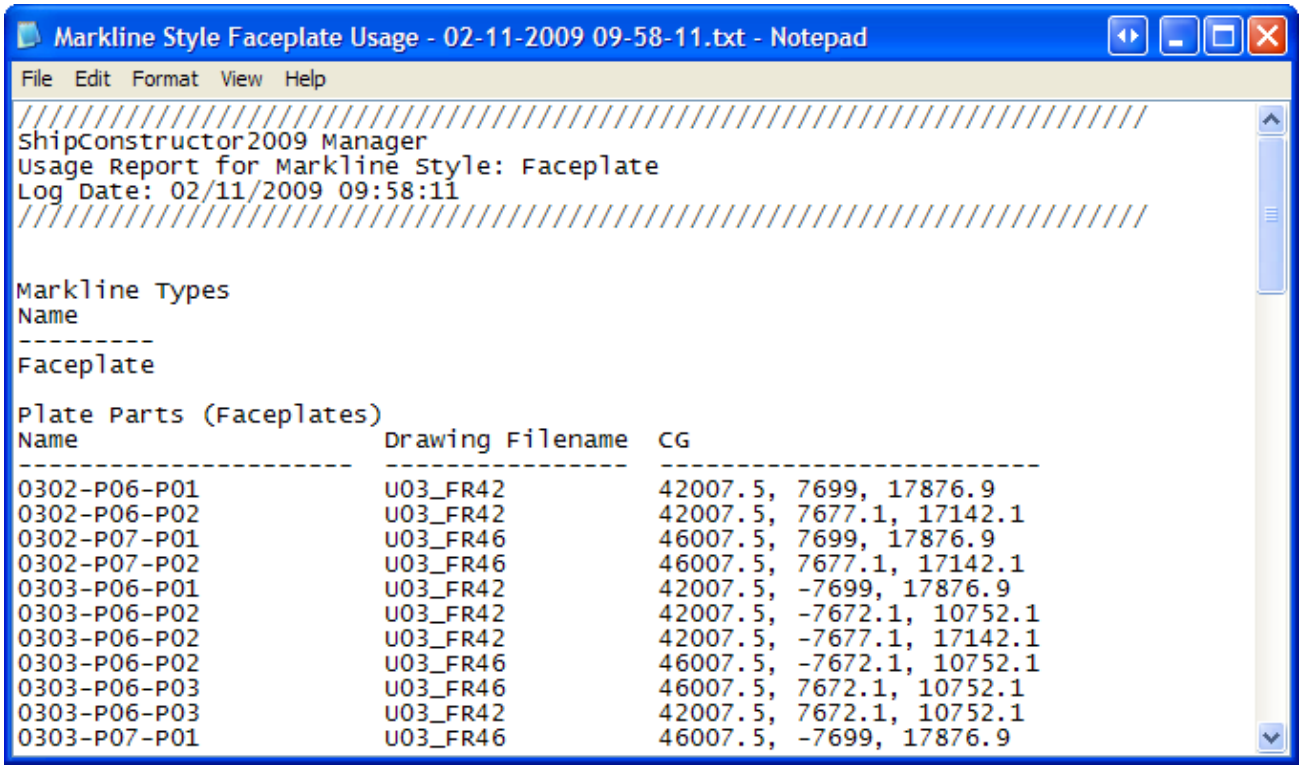
View Markline Style Usage

It is possible to see a list of all the parts that use a selected markline style. A check mark in the In Use column indicates that the markline style is being used by parts.

To list all parts using a specific markline style

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Markline Styles (page 348) to open the Markline Styles window.
3. Select the markline style you want.
4. Click Usage Log.

   ShipConstructor displays a list of all parts using the selected markline style.



Set Up Markline Symbols

Markline symbols are used on marklines to generate special symbols. Markline symbols let you customize all generated symbols.

When setting up symbols, it is recommended to draw them to a nominal size 1. This lets you specify a size in the markline as the actual size you want for production.

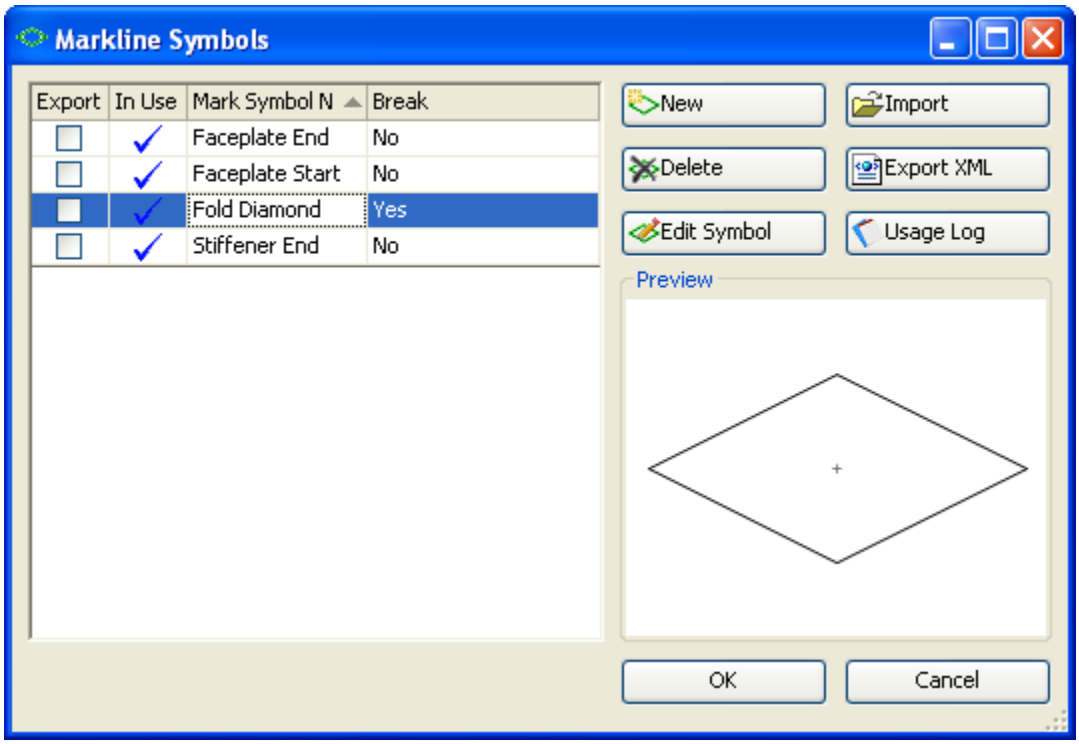
Valid objects for markline symbols are lines, arcs, circles, and polylines.

When you create a markline symbol, the (0,0) point in the drawing is the reference point used when adding it to a markline. If the symbol is intended to be at the end of a markline, draw the symbol in the positive X direction. When the symbol is applied to the end of the markline, the symbol is mirrored. For stiffener marklines, the throw direction of the stiffener is in the positive Y direction.

Create a Markline Symbol

To create a markline symbol

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Markline Styles (page 348) to open the Markline Styles window.
3. Click Edit Symbols to open the Markline Symbols window.



4. Click New.
5. Type a name for the markline symbol and press Enter.
6. To break the markline where each symbol is inserted, set Break Markline to Yes.
7. Click Edit Symbol.

   ShipConstructor opens a blank drawing.

   Note: A window named Save Mark Symbol appears in the top left corner of the screen.

8. Use AutoCAD's drawing tools to create the symbol out of lines, arcs, polylines, or ellipses.
9. Click Save in the Save Mark Symbol window.

   ShipConstructor returns to the Markline Symbols window.
10. Click OK to close the Markline Symbols window.
11. Click OK to close the Markline Styles window.

Edit a Markline Symbol

To edit a markline symbol

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Markline Styles (page 348) to open the Markline Styles window.
3. Click Edit Symbols to open the Markline Symbols window.
4. Select the markline symbol you want to edit.

5. Click Edit Symbol.

   ShipConstructor opens the symbol's drawing.

   Note: A window named Save Mark Symbol appears in the top left corner of the screen.

6. Use AutoCAD's drawing tools to edit the symbol.
7. Click Save in the Save Mark Symbol window.

   ShipConstructor returns to the Markline Styles window.
8. Click OK to close the Markline Styles window.

Delete a Markline Symbol

To delete a markline symbol

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Markline Styles (page 348) to open the Markline Styles window.
3. Click Edit Symbols to open the Markline Symbols window.
4. Select the markline symbol to delete.
5. Click Delete.
6. Click OK to close the Markline Symbols window.
7. Click OK to close the Markline Styles window.

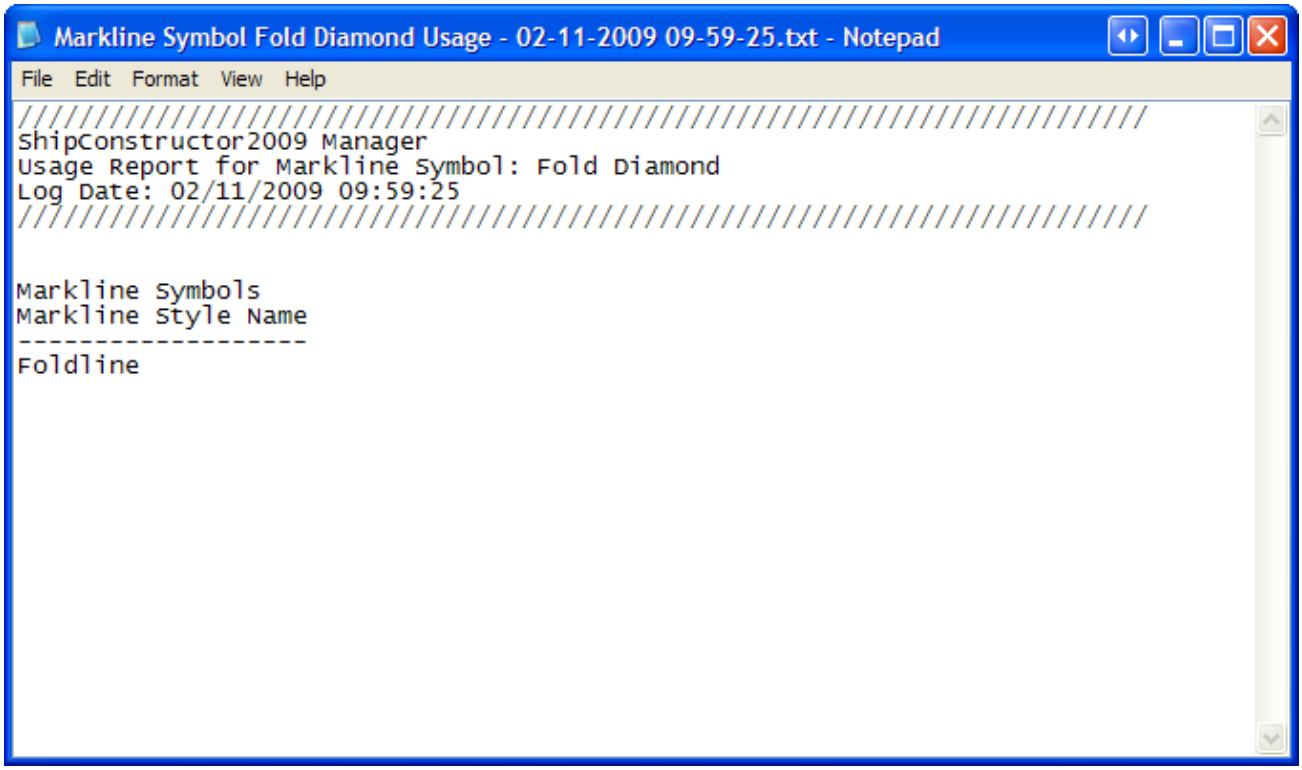
View Markline Symbol Usage

It is possible to see a list of all the markline styles that use a selected markline symbol. A check mark in the In Use column indicates that the markline symbol is being used by markline styles.

To list all markline styles using a specific markline symbol

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Markline Styles (page 348) to open the Markline Styles window.
3. Click Edit Symbols to open the Markline Symbols window.
4. Select the markline symbol you want.
5. Click Usage Log.

   ShipConstructor displays a list of all markline styles using the selected markline symbol.

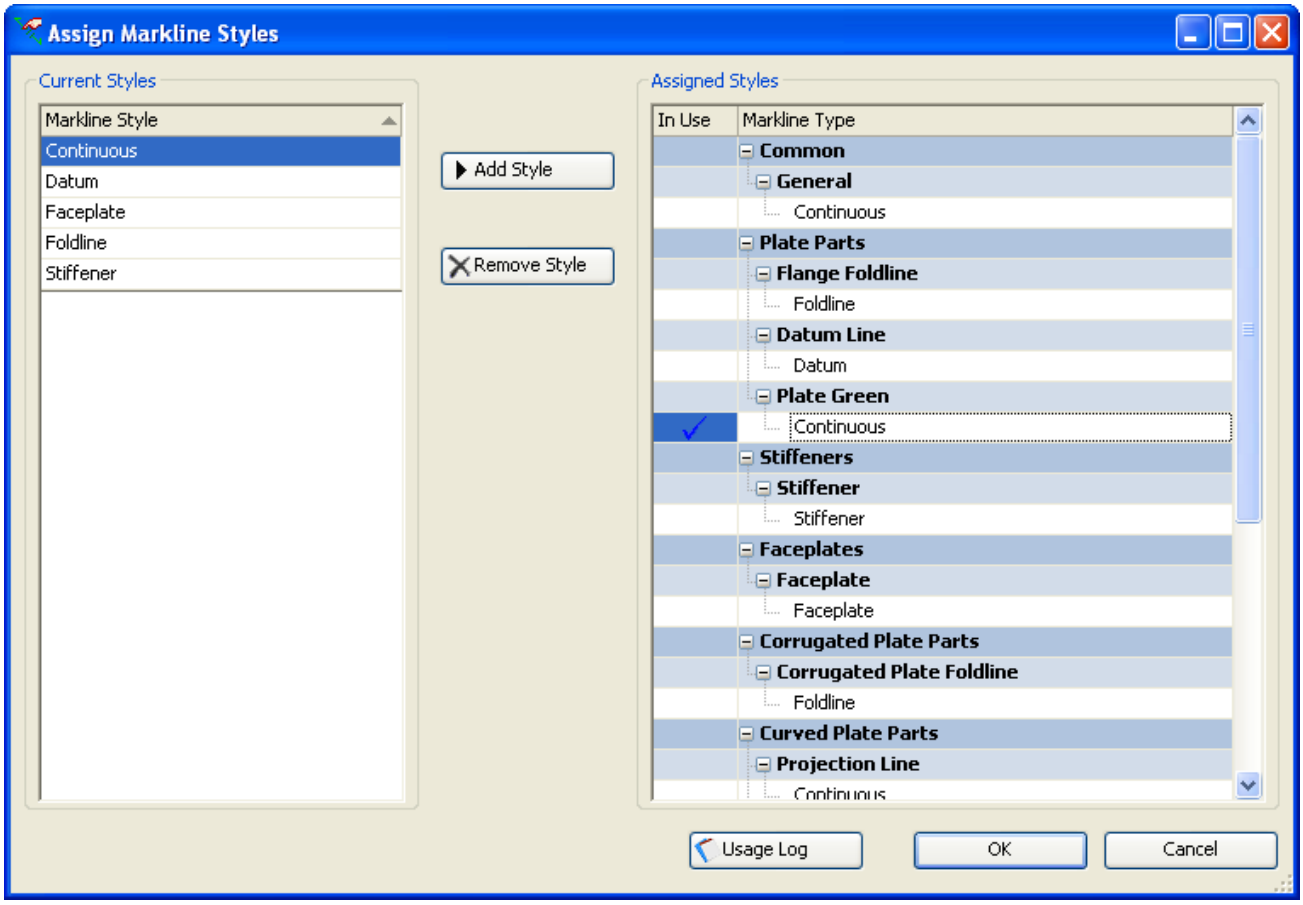

Markline Symbol Fold Diamond Usage - 02-11-2009 09-59-25.txt - Notepad
File Edit Format View Help
ShipConstructor2009 Manager
Usage Report for Markline Symbol: Fold Diamond
Log Date: 02/11/2009 09:59:25
Markline Symbols
Markline Style Name
Foldline

Assign Markline Styles

You can control which markline styles can be used for stiffeners, flange foldlines, corrugated plate foldlines, and datum lines.

To assign markline styles

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Markline Styles (page 348) to open the Markline Styles window.
3. Click Assign Styles to open the Assign Markline Styles window.



4. Under Assigned Styles, select a Markline Type.
5. Under Current Styles, select a Markline Style.
6. Click Add Style to add the selected markline style to the selected markline type.
7. Continue adding markline styles to markline types.
8. Click OK to close the Assign Markline Styles window.
9. Click OK to close the Markline Styles window.

View Markline Style Usage

It is possible to see a list of all the parts that use a markline style of a markline type. A check mark in the In Use column indicates that the markline style of a markline type is being used by parts.

To list all parts using a specific markline style of a markline type

5. Choose ShipConstructor > Manager (page 383) to open Manager.
6. Choose Structure > Plates > Markline Styles (page 348) to open the Markline Styles window.
7. Click Assign Styles to open the Assign Markline Styles window.
8. Select the markline style of a markline type that you want.
9. Click Usage Log.

   ShipConstructor displays a list of all parts using the selected markline style of a markline type.

Markline Type Flange Foldline - Markline Style Foldline Usage - 02-11-2009 10-01-55.txt

File Edit Format View Help

```
////////////////////////////////////////////////////////////////////////////////
ShipConstructor2009 Manager
Usage Report for Markline Style Foldline assigned to Markline Type Flange
Foldline:
Log Date: 02/11/2009 10:01:55
////////////////////////////////////////////////////////////////////////////////


Plate Parts (Flanges)
Name                    Drawing Filename  CG
----------------------  ----------------  -------------------------
0301-P07-P02            U03_FR39          38984.9, 7640.7, 1519.8
0301-P07-P13            U03_FR40          39984.9, 7640.5, 1488.8
0301-P07-P15            U03_FR41          40984.9, 7641.1, 1467.3
0301-P07-P18            U03_FR43          42985, 7642, 1436.7
0301-P07-P24            U03_FR44          43985, 7642.4, 1426.4
0301-P07-P30            U03_FR47          46985.1, 7643.5, 1410
0301-P07-P34            U03_FR48          47985.1, 7643.8, 1408
0301-P07-P36            U03_FR49          48984.9, 7641.9, 1417.5
0301-P07-P40            U03_FR45          44985, 7642.8, 1418.7
0301-P08-P04            U03_FR41          40984.9, -7641.1, 1467.3
0301-P08-P06            U03_FR43          42985, -7642, 1436.7
0301-P08-P08            U03_FR40          39984.9, -7640.5, 1488.8
0301-P08-P10            U03_FR39          38999.9, -7640.6, 1518.9
0301-P08-P14            U03_FR44          43985, -7642.4, 1426.4
0301-P08-P20            U03_FR47          46985.1, -7643.5, 1410
```

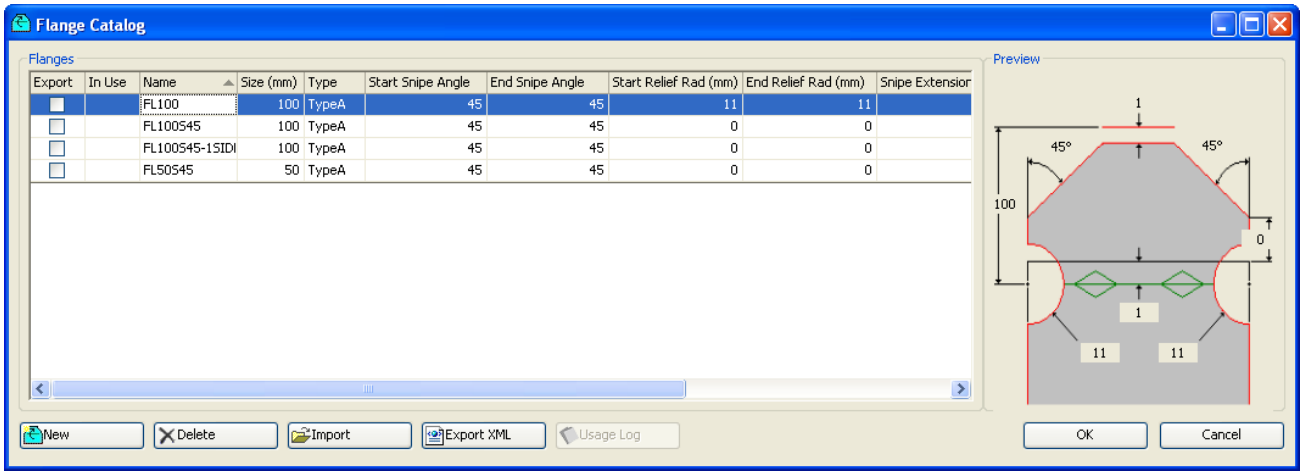
Flange Catalog

A flange is the result of bending the edge of a plate. A flange is intended to strengthen the plate or attach the plate to another object. You must predefine the various types of flanges you plan to use in your project in Manager. During structural modeling, you can then insert a flange onto a plate by selecting it from the list of pre-defined flanges.

Create a Flange Standard

To create a flange

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Flange Catalog (page 352) to open the Flange Catalog window.



3. Click New.
4. Enter a name for the flange and press Enter.
5. Set the values for the flange. See Flange Library Window (page 353).

   When you select a field to edit, the corresponding value becomes highlighted in the image on the right of the window.
6. Click OK to close the Flange Catalog window.

The bend radius of the flange for production is determined by the flange and foldline corrections. The bend radius for the solid is set from the Flg Inner Rad (X Thick) column in the Plate Stock Editor. This can be overridden for each instance.

Edit a Flange

To edit a flange

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Flange Catalog (page 352) to open the Flange Catalog window.
3. Click the field of the flange you want to edit.
4. Enter a new value or setting.
5. Press Enter

Delete a Flange

To delete a flange

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Flange Catalog (page 352) to open the Flange Catalog window.
3. Select the flange you want to delete.
4. Click Delete.

Import Flanges

To import flanges

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Flange Catalog (page 352) to open the Flange Catalog window.
3. Click Import to open a File Browser.
4. Select a project file (*.PRO) or XML file (*.XML).
5. Click Open.

Export Flanges

To export flanges

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Flange Catalog (page 352) to open the Flange Catalog window.
3. Check the Export check boxes for the flanges to export.
4. Click Export XML to open a File Browser.
5. Enter a name for the XML file.
6. Click Save.

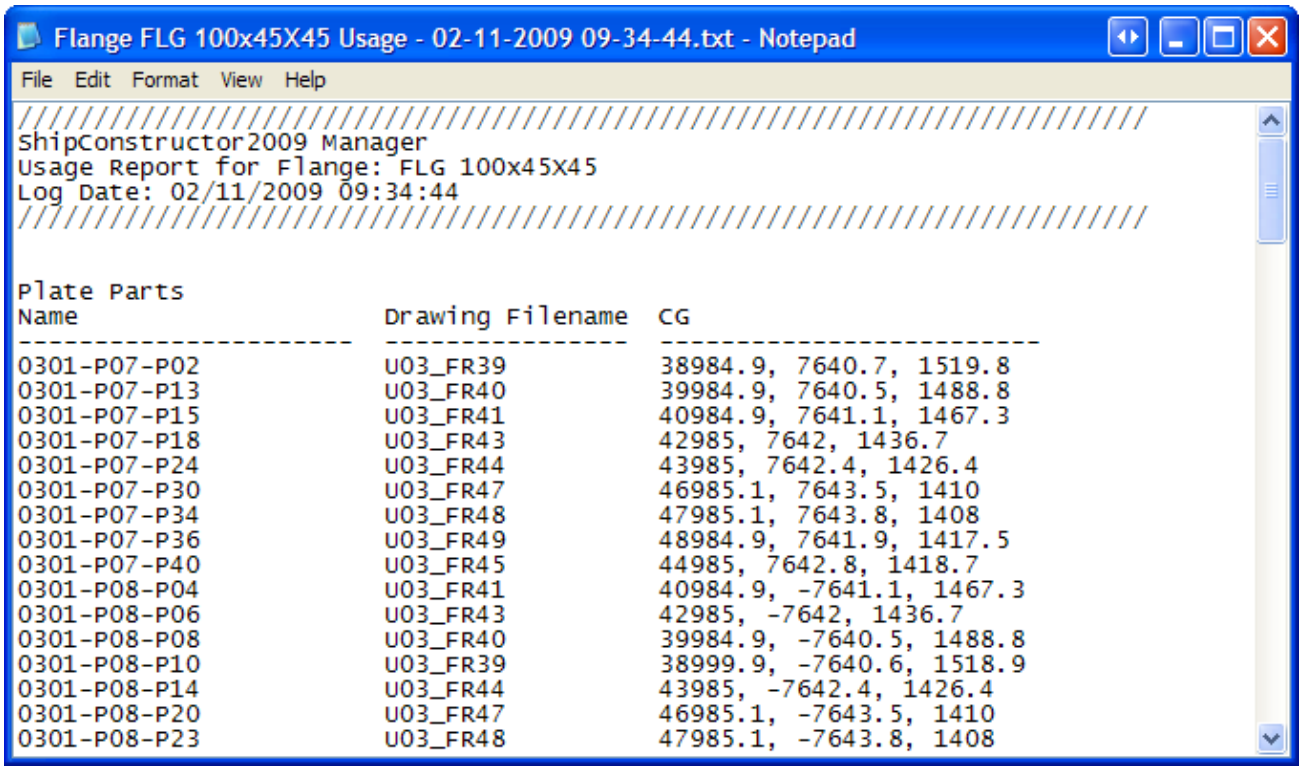
View Flange Usage

It is possible to see a list of all the parts that use a selected flange. A check mark in the In Use column indicates that the flange is being used by parts.

To list all parts using a specific flange

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Flange Catalog (page 352) to open the Flange Catalog window.
3. Select the flange you want.
4. Click Usage Log.

   ShipConstructor displays a list of all parts using the selected flange.



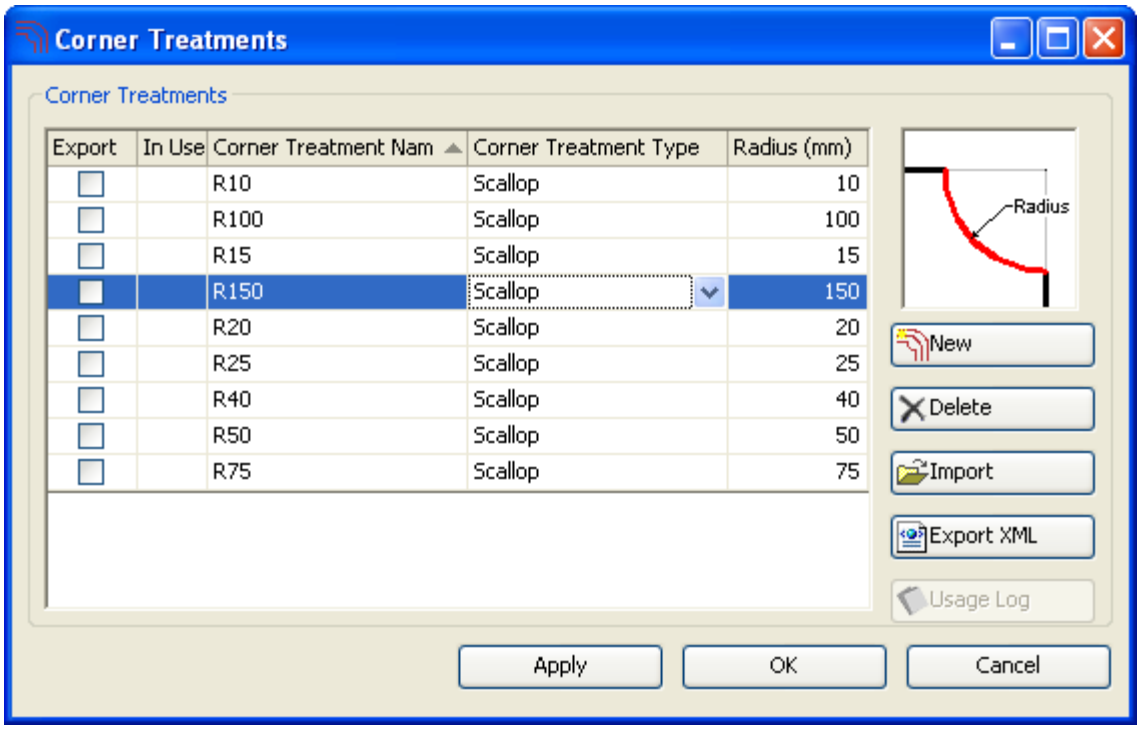
Corner Treatments

Corner treatments define different types of scallops, fillets, and snipes that you can apply to plate parts during detailing. Changing the corner treatment standard changes all instances of the corner treatments.

Create a Corner Treatment

To create a corner treatment

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Corner Treatments (page 355) to open the Corner Treatments window.



3. Click New.
4. Enter a name for the corner treatment and press Enter.
5. Select the Corner Treatment Type: Scallop, Fillet, or Snipe.
6. Set the corner treatment Radius.
7. Click Apply to save the changes, or click OK to save the changes and close the Corner Treatments window.

Delete a Corner Treatment

To delete a corner treatment

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Corner Treatments (page 355) to open the Corner Treatments window.
3. Select the corner treatment.
4. Click Delete.
5. Click Apply to save the changes, or click OK to save the changes and close the Corner Treatments window.

View Corner Treatment Usage

It is possible to see a list of all the parts that use a selected corner treatment. A check mark in the In Use column indicates that the corner treatment is being used by parts.

To list all parts using a specific corner treatment

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Corner Treatments (page 355) to open the Corner Treatments window.
3. Select the corner treatment you want.
4. Click Usage Log.

   ShipConstructor displays a list of all parts using the selected corner treatment.

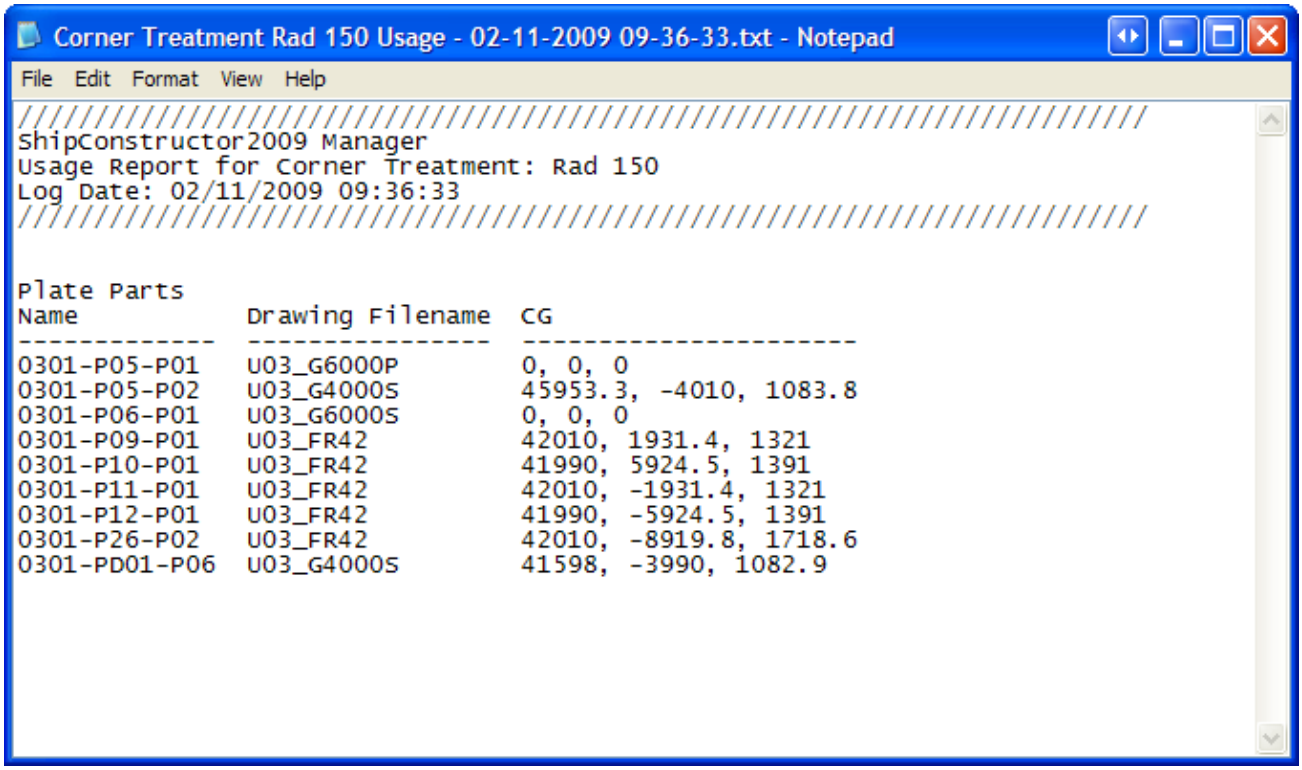


Plate Green Standards

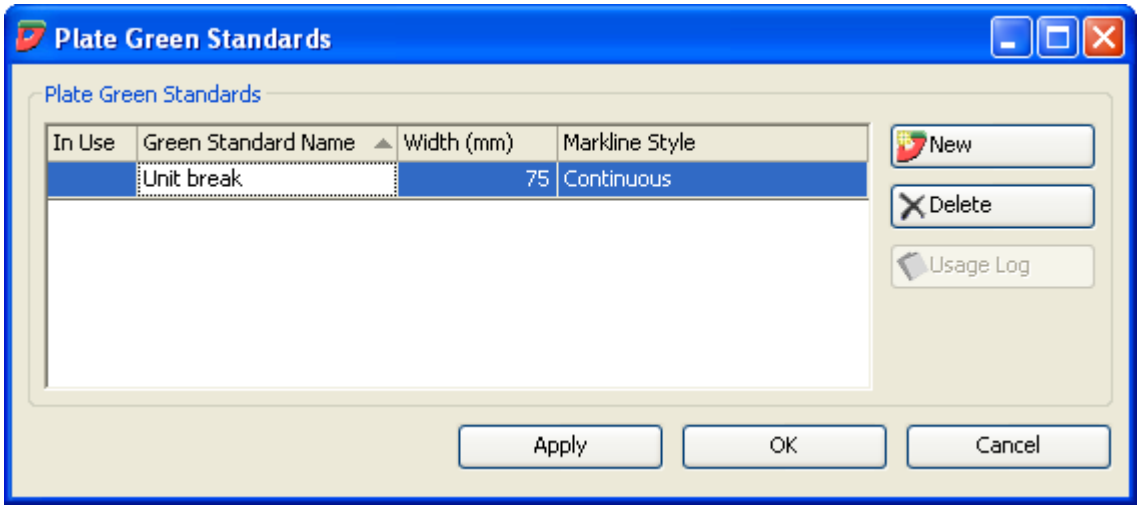
You can now define one or more standards for plate green (for example, unit break green and assembly green). These can then be added to plate parts in a parametric fashion. You can later change the value (the size of the extra material), and all parts using the standard will be updated.

Note: There must be a markline style defined and assigned to plate green before you can open this window.

Create a Plate Green Standard

To create a plate green standard

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Green Standards (page 355) to open the Plate Green Standards window.



3. Click New.
4. Enter a name for the plate green standard and press Enter.
5. Set the Width.
6. Select the Markline Style.

Click Apply to save the changes, or click OK to save the changes and close the Plate Green Standards window.

View Plate Green Standard Usage

It is possible to see a list of all the parts that use a selected plate green standard. A check mark in the In Use column indicates that the plate green standard is being used by parts.

To list all parts using a specific plate green standard

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Green Standards (page 355) to open the Plate Green Standards window.
3. Select the plate green standard you want.
4. Click Usage Log.

ShipConstructor displays a list of all parts using the selected plate green standard.

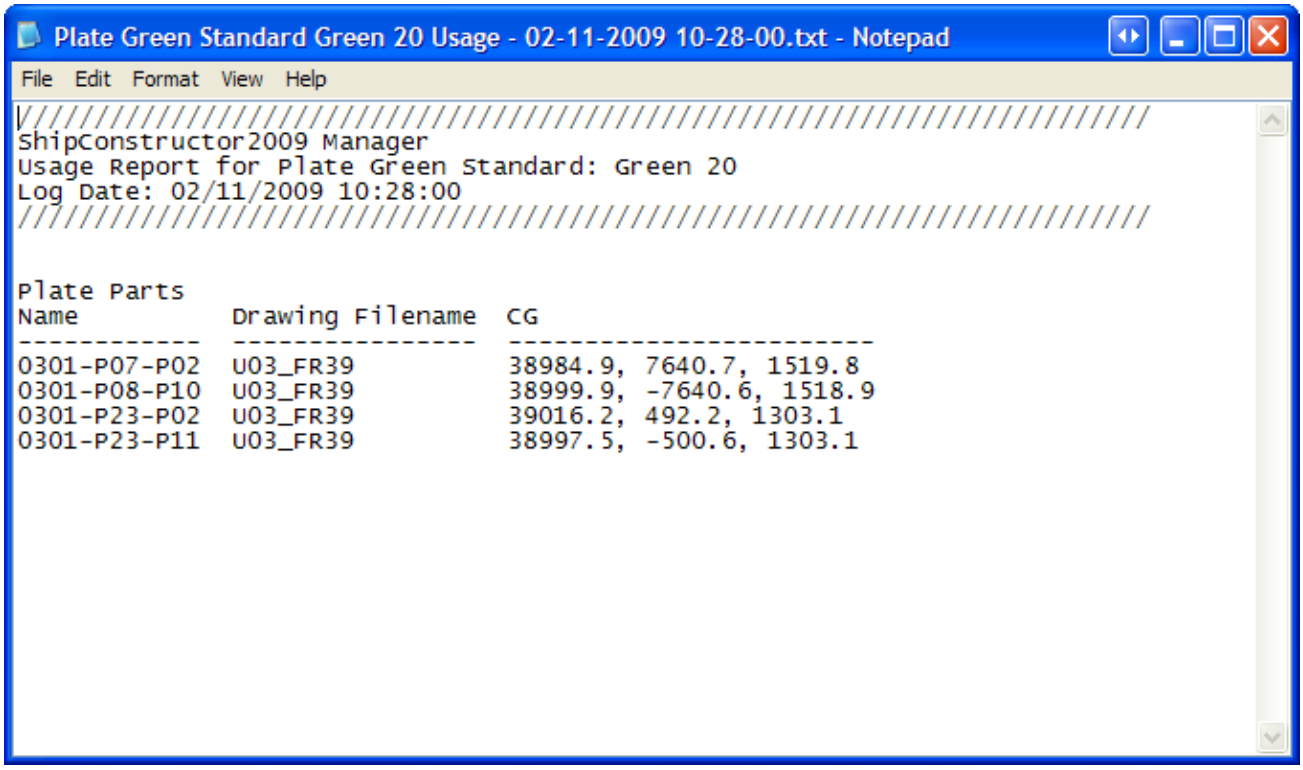


Plate Green Standard Green 20 Usage - 02-11-2009 10-28-00.txt - Notepad
File Edit Format View Help
//
ShipConstructor2009 Manager
Usage Report for Plate Green Standard: Green 20
Log Date: 02/11/2009 10:28:00
//
Plate Parts
Name Drawing Filename CG
0301-P07-P02 U03_FR39 38984.9, 7640.7, 1519.8
0301-P08-P10 U03_FR39 38999.9, -7640.6, 1518.9
0301-P23-P02 U03_FR39 39016.2, 492.2, 1303.1
0301-P23-P11 U03_FR39 38997.5, -500.6, 1303.1

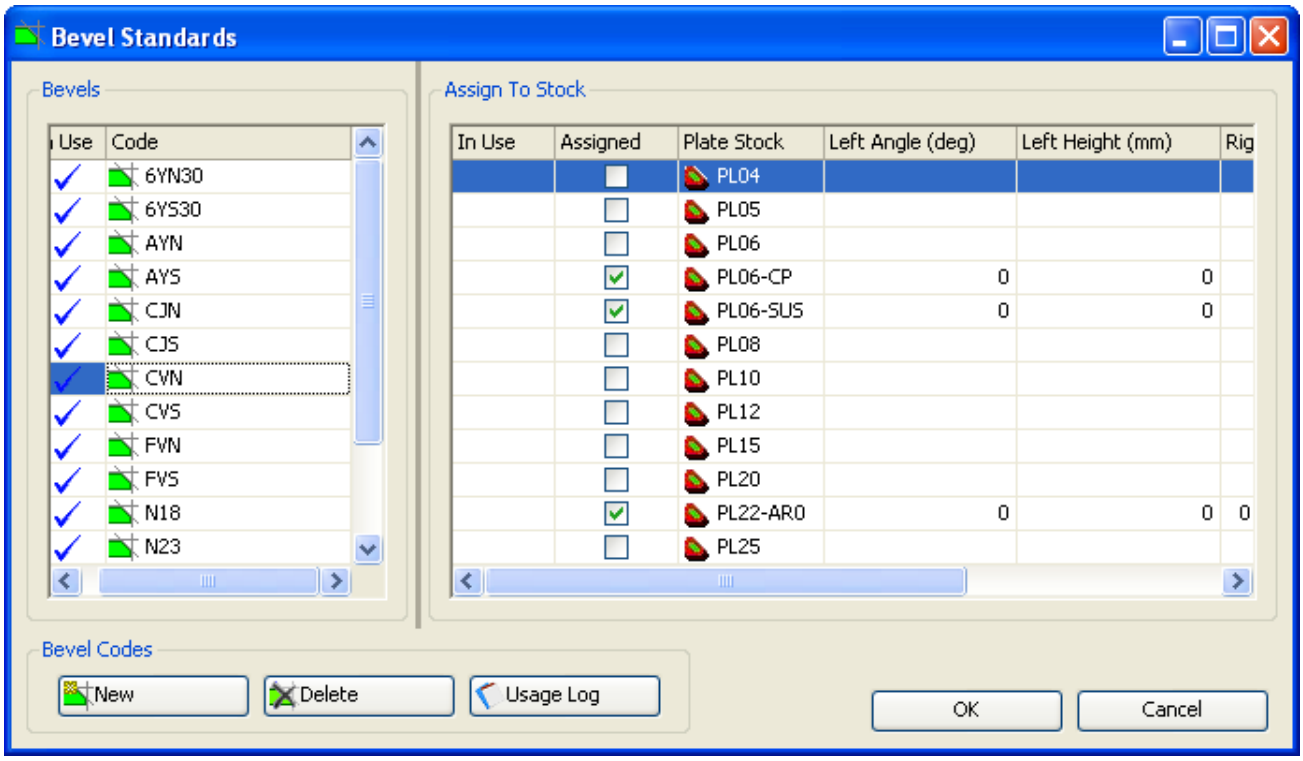
Bevel Standards

A bevel standard is a definition of a bevel that can be used by NC-Pyros to generate bevel cutting NC codes.

Create a Bevel Standard

To create a bevel standard

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Bevel Standards (page 357) to open the Bevel Standards window.



3. Click New to create a new bevel code.
4. Assign the bevel code to plate stocks as follows:

   Under Bevels, select the bevel code to assign to plate stocks. Under Assign to Stock, click the check boxes for the plate stocks to assign the bevel code to.
5. Under Assign to Stock, enter the values for Left Angle, Left Height, Right Angle, and Right Height. (Left and right refer to the two outer torches of a three torch head.)
6. Click OK to close the Bevel Standards window.

View Bevel Standard Usage

It is possible to see a list of all the parts that use a selected bevel standard. A check mark in the In Use column indicates that the bevel standard is being used by parts.

To list all parts using a specific bevel standard

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Bevel Standards (page 357) to open the Bevel Standards window.
3. Select the bevel standard you want.
4. Click Usage Log.

   ShipConstructor displays a list of all parts using the selected bevel standard.

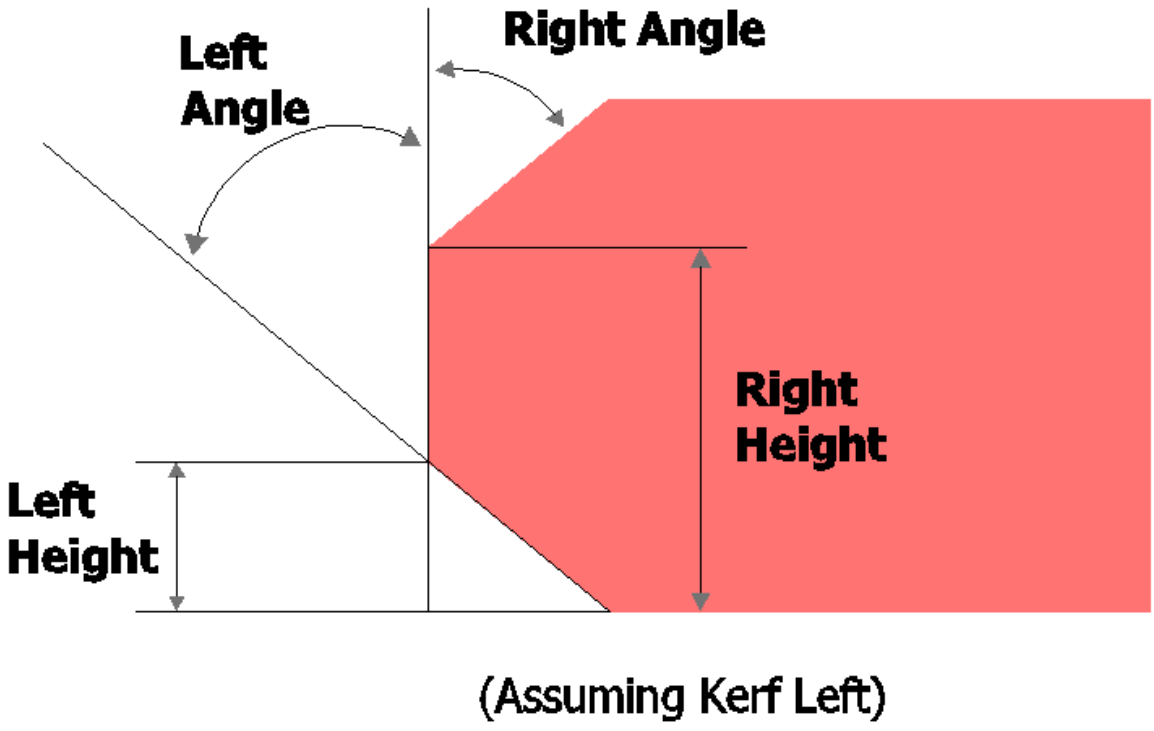
Bevel Code AK15 Usage - 02-11-2009 10-32-00.txt - Notepad

File Edit Format View Help

```
////////////////////////////////////////////////////////////////////////////////
ShipConstructor2009 Manager
Usage Report for Bevel Code: AK15
Log Date: 02/11/2009 10:32:00
////////////////////////////////////////////////////////////////////////////////


Plate Parts
Name            Drawing Filename  CG
--------------  ----------------  -------------------------
0302-P04-P02    U03_FR39          38985.1, 9250, 11670.6
0302-PD03-P02   U03_FR39          38985.9, 9209.4, 17907.1
0303-P04-P02    U03_FR39          38985.1, -9250, 11670.6
0303-PD03-P02   U03_FR39          38985.9, -9209.4, 17907.1
```

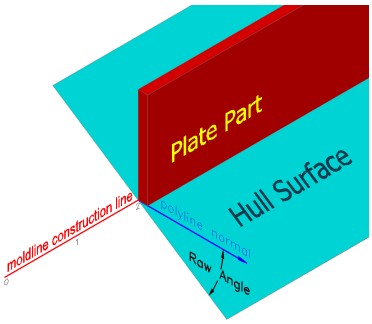
Left and Right Explained

The following view is of a plate with a bevel lying on the NC table. This would be the case when the torch is traveling into the screen.



Automatic Bevels

ShipConstructor can add bevels to plate parts automatically. Automatic beveling will add a simple bevel cut to plate parts based on the angle of the hull surface adjacent to the plate. Bevel angles are calculated automatically and stored with constructionlines. The angle is determined from the angle of the hull surface at each point on the line, as shown here:



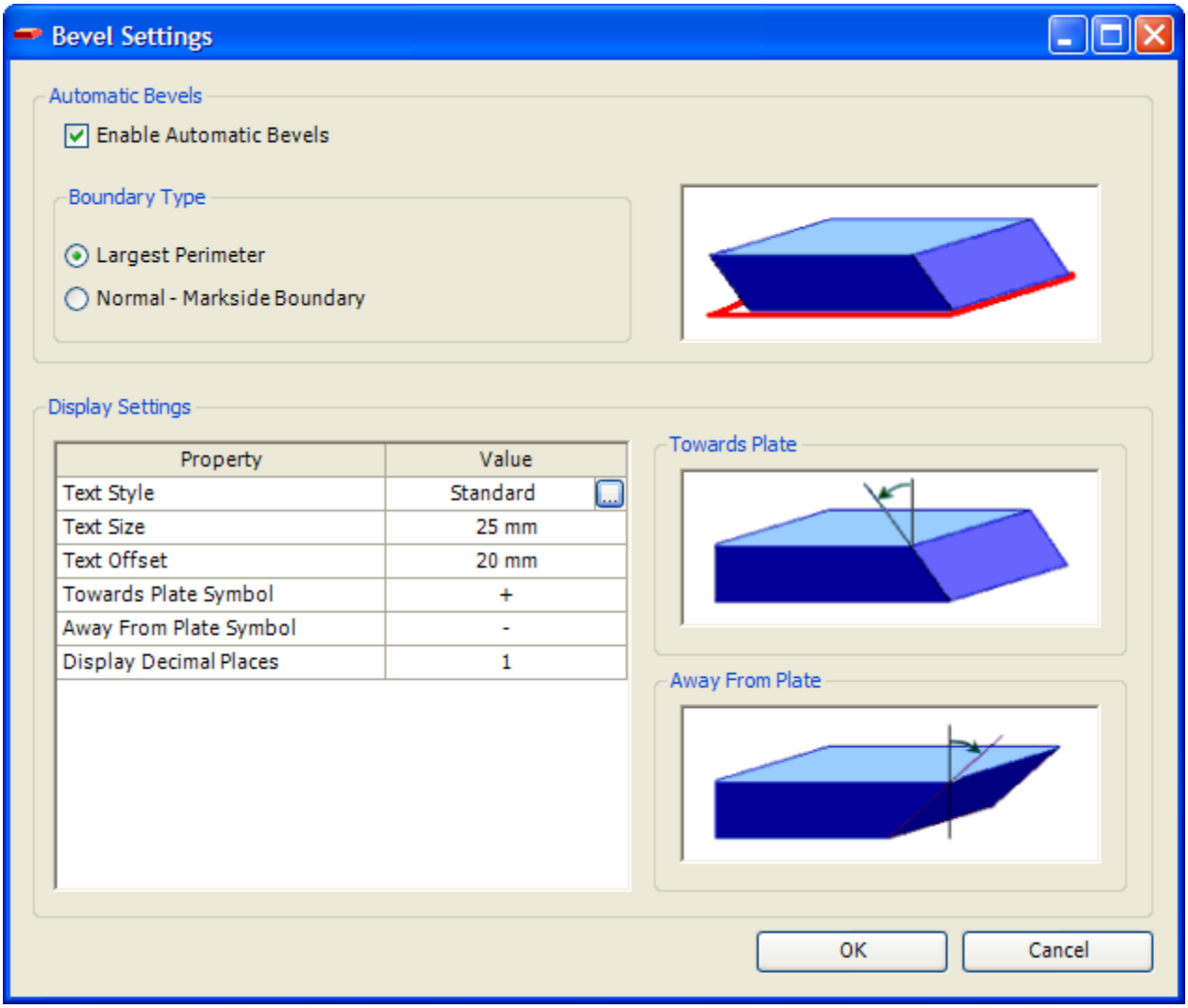
If the moldline constructionline shown here is used to create a plate part, bevels will be added to the part from the raw angles stored on the construction line. Bevels are created dynamically and updated with the constructionline – if the angles change, the bevels on the part will be updated to reflect the new angles.

Automatic beveling is configured through the Bevel Settings dialog.

To configure automatic beveling:

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > Bevel Settings (page 357) to open the Bevel Settings window.

The Bevel Settings window is displayed. The configuration settings are explained below.



Enable Automatic Bevels – Turns automatic bevels on or off. When this checkbox is checked, bevels will be added to all plates that have bevel angles on their construction lines, including plates that have already been created.

Boundary Type – Selects the type of boundary to create when adding automatic bevels. Largest perimeter will create a boundary from the farthest outside path around the part. This may be the markside outline, non-markside outline, or a combination of both. Normal – Markside will always use the markside outline. The boundary type will depend on the machine used to cut the plates.

Display Settings – These settings affect how the bevel information is displayed in the model drawing and how it is exported to NCPyros:

Text Style – The text style used in the model drawing used when displaying bevels.

Text Size – The size of the text used to display bevels.

Text Offset – The distance from the edge of the plate to the bevel text.

Towards Plate Symbol – Symbol to display in front of text for bevels that are cut towards the plate.

Away From Plate Symbol – Symbol to display in front of text for bevels that are cut away from plate.

Export Decimal Places – Number of decimal places to use when exporting bevel information to NC Pyros.

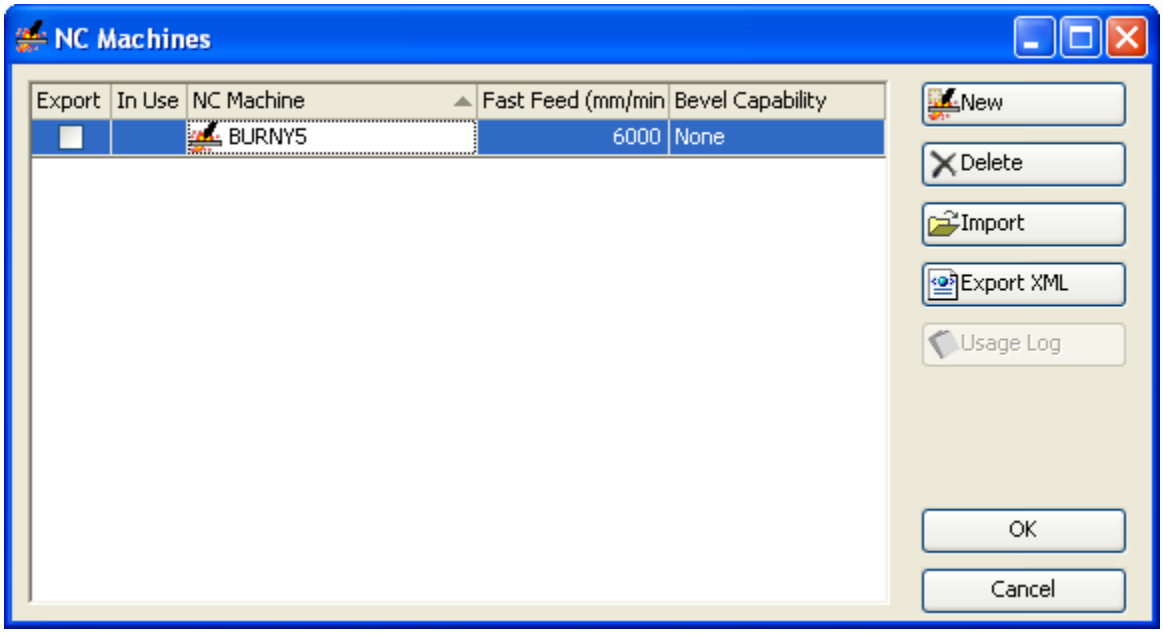
NC Machines

NC machines cut plates from plate stock. You must set up your NC machines before plate nesting occurs. See Plate Nesting (page 254).

Create an NC Machine

To create an NC machine

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > NC Machines (page 356) to open the NC Machines window.



3. Click New.
4. Enter a name for the NC machine and press Enter.
5. Set the Fast Feed (fast travel feed rate). See NC Machines Window (page 357).
6. Click OK to close the NC Machines window.

Edit an NC Machine

To edit an NC machine

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > NC Machines (page 356) to open the NC Machines window.
3. Double-click the field you want to edit, enter the new value you want, and press Enter.
4. Click OK to close the NC Machines window.

Delete an NC Machine

To delete an NC machine

Note: You cannot delete an NC machine that has been assigned to a plate. You must first delete any plates that are using the NC machine.

1. Choose ShipConstructor > Manager (page 383) to open Manager.

2. Choose Structure > Plates > NC Machines (page 356) to open the NC Machines window.
3. Select the NC machine.
4. Click Delete.
5. Click OK to close the NC Machines window.

Import NC Machines

To import NC machines

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > NC Machines (page 356) to open the NC Machines window.
3. Click Import to open a File Browser.
4. Select a project file (*.PRO) or XML file (*.XML).
5. Click Open.

Export NC Machines

To export NC machines

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > NC Machines (page 356) to open the NC Machines window.
3. Check the Export check boxes for the NC machines to export.
4. Click Export XML to open a File Browser.
5. Enter a name for the XML file.
6. Click Save.

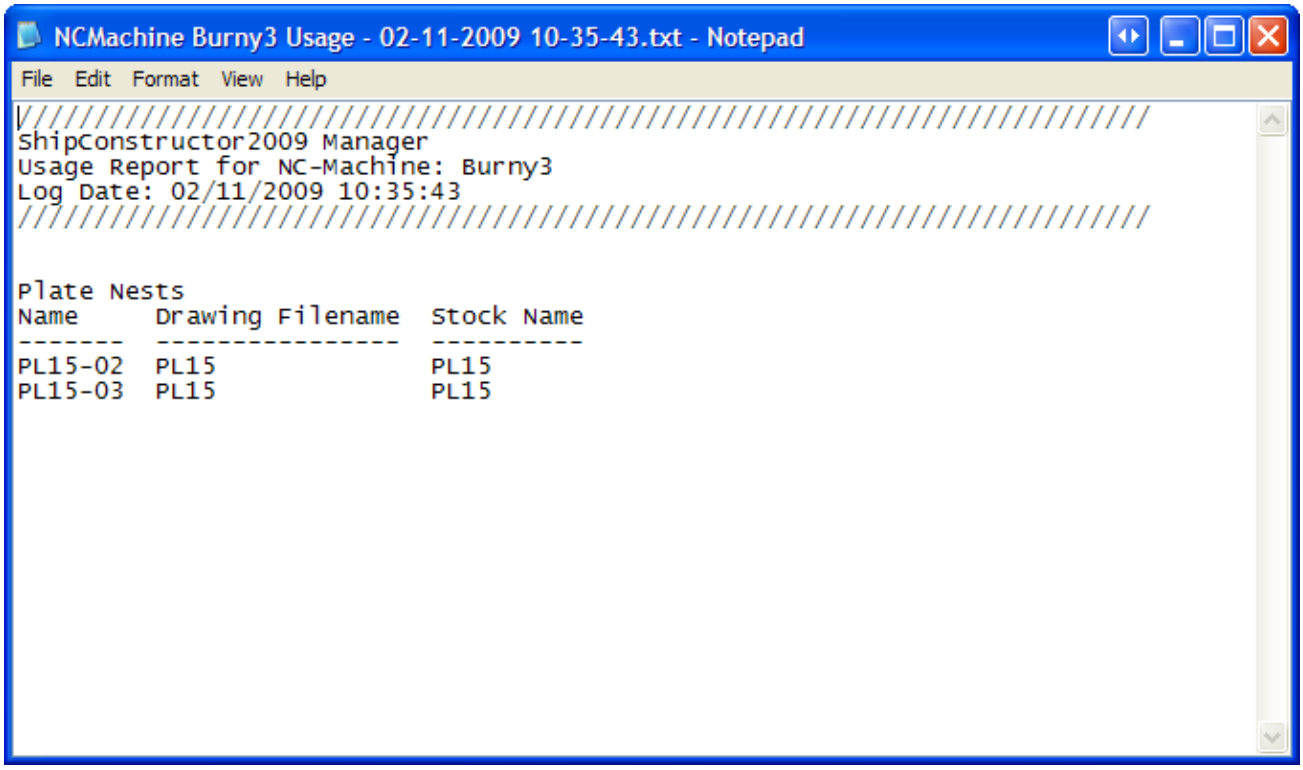
View NC Machine Usage

It is possible to see a list of all the plate nests that use a selected NC machine. A check mark in the In Use column indicates that the NC machine is being used by plate nests.

To list all plate nests using a specific NC machine

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Plates > NC Machines (page 356) to open the NC Machines window.
3. Select the NC machine you want.
4. Click Usage Log.

   ShipConstructor displays a list of all plate nests using the selected NC machine.



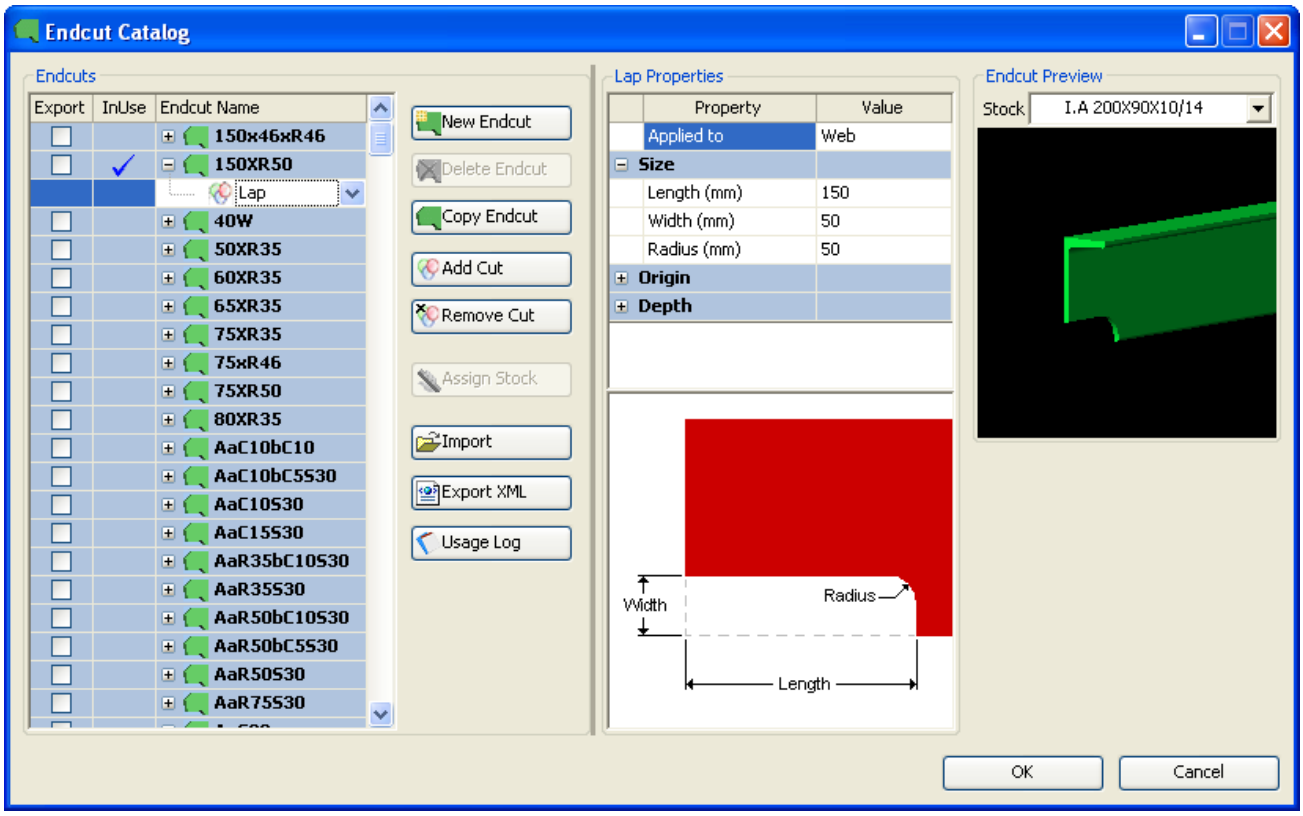
Endcuts

An endcut defines how profile stocks can potentially be cut at their end. You can set up several different types of endcuts and then apply them to individual stiffener parts as necessary. The end of a stiffener can have endcuts and/or trims applied. Endcuts are predefined shapes, whereas trims are plane section cuts that can be at any angle to the end. Trims are applied on a per instance basis.You can adjust the endcut's position when the stiffener's end is also trimmed.

Create an Endcut

To create an endcut

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Profiles > Endcut Catalog (page 358) to open the Endcut Catalog window.



3. Click New Endcut.
4. Enter a name for the endcut and press Enter.
5. Click Assign Stock to open the Endcut Assignment window.
6. Check the check box for each stock that you want to use the endcut with.
7. Click OK to close the Endcut Assigment window.
8. Set the Endcut Preview Stock to one of the assigned stocks to display a preview of the endcut.

   The preview shows the overall effect of the endcut on the selected profile stock (that is, even if the endcut consists of several cuts).

   One endcut is composed of one or more cuts.
9. Click Add Cut to add a cut to the endcut.
10. Select the cut type (for example, Radius, Rectangle, or Triangle).
11. Select whether the cut is Applied to the Web or Flange of the profile stock.
12. Specify the size of the cut by setting the Size values.

    Note: The Size values are different for each cut type.

    An image below each listed properties indicates the property's definition.

    To specify a constant value, enter the value. To specify a variable value (based on the web or flange dimensions of the profile stock), click to open the Endcut Formula window, and enter a formula.
13. Specify the position of the cut by setting the Origin values.

    Note: When you select or edit an Origin value, the image below changes to indicate the effect of the values.

To specify a constant value, enter the value. To specify a variable value (based on the web or flange dimensions of the profile stock), click to open the Endcut Formula window and enter a formula.Select one of the Trim Adjustment options to reposition the endcut after trimming or select None to place the endcut before trim(s).

14. Specify the depth of the cut by setting the Depth values.

    Note: When you select or edit a Depth value, the image below changes to indicate the effect of the values.

    To specify a constant value, enter the value. To specify a variable value (based on the web or flange dimensions of the profile stock), click to open the Endcut Formula window, and enter a formula.
15. If necessary, add additional cuts to the endcut.
16. Click OK to close the Endcut Definitions window.

Delete an Endcut

To delete an endcut

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Profiles > Endcut Catalog (page 358) to open the Endcut Catalog window.
3. Select the endcut.
4. Click Delete Endcut.

Import Endcuts

To import endcuts

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Profiles > Endcut Catalog (page 358) to open the Endcut Catalog window.
3. Click Import to open a File Browser.
4. Select a project file (*.PRO) or XML file (*.XML).
5. Click Open.

Export Endcuts

To export endcuts

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Profiles > Endcut Catalog (page 358) to open the Endcut Catalog window.
3. Check the Export check boxes for the endcuts you want to export.
4. Click Export XML to open a File Browser.
5. Enter a name for the XML file.
6. Click Save.

View Endcut Usage

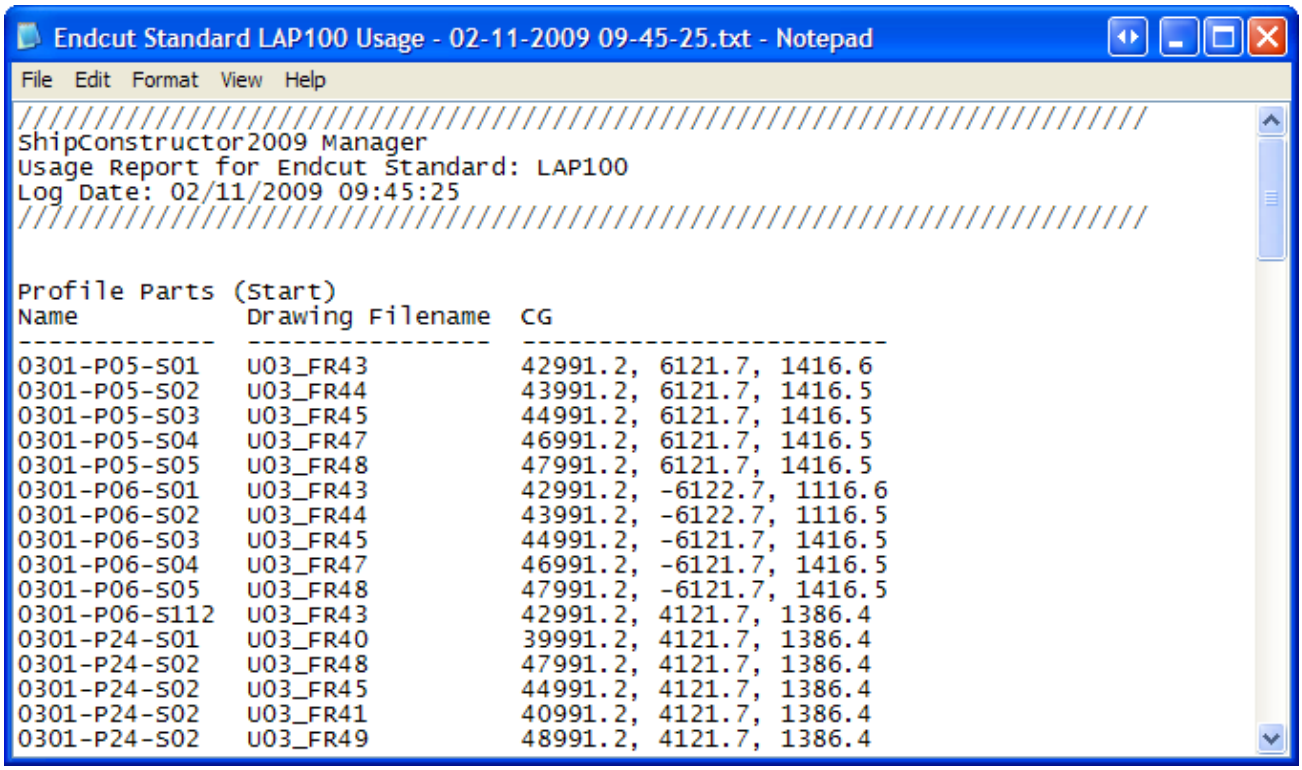
It is possible to see a list of all the parts that use a selected endcut. A check mark in the In Use column indicates that the endcut is being used by parts.

To list all parts using a specific endcut

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Profiles > Endcut Catalog (page 358) to open the Endcut Catalog window.
3. Select the endcut you want.

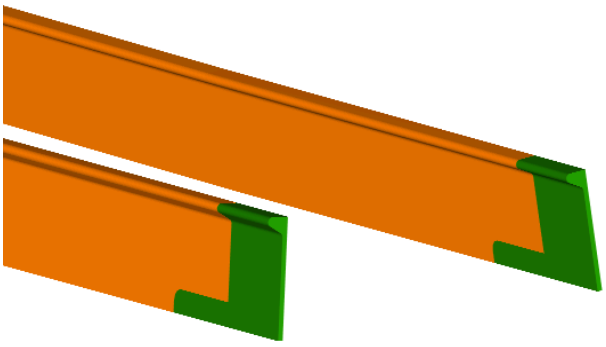
4. Click Usage Log.

   ShipConstructor displays a list of all parts using the selected endcut.



Profile Green Standards

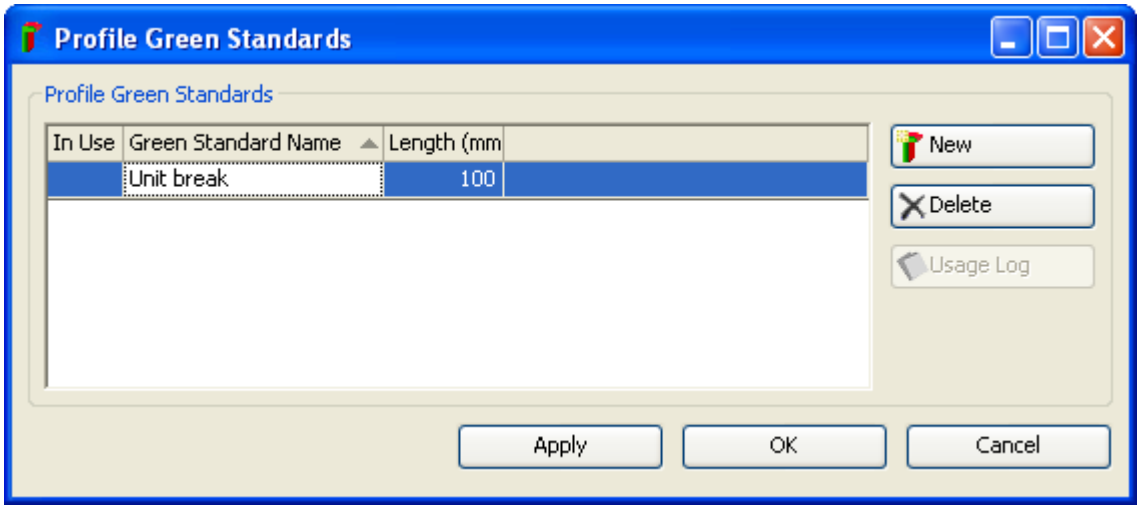
Like plate green, profile green is the extra stock or extra material used wherever you have to join assemblies together and need extra stock to compensate for tolerances in production. You can now define one or more standards for profile green that can then be added to profile parts in a parametric fashion. You can later change the value (the size of the extra material), and all parts using the standard will be updated.



Create a Profile Green Standard

To create a profile green standard

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Profiles > Green Standards (page 361) to open the Profile Green Standards window.



3. Click New.
4. Enter a name for the profile green standard and press Enter.
5. Set the Length.
6. Click Apply to save the changes, or click OK to save the changes and close the Profile Green Standards window.

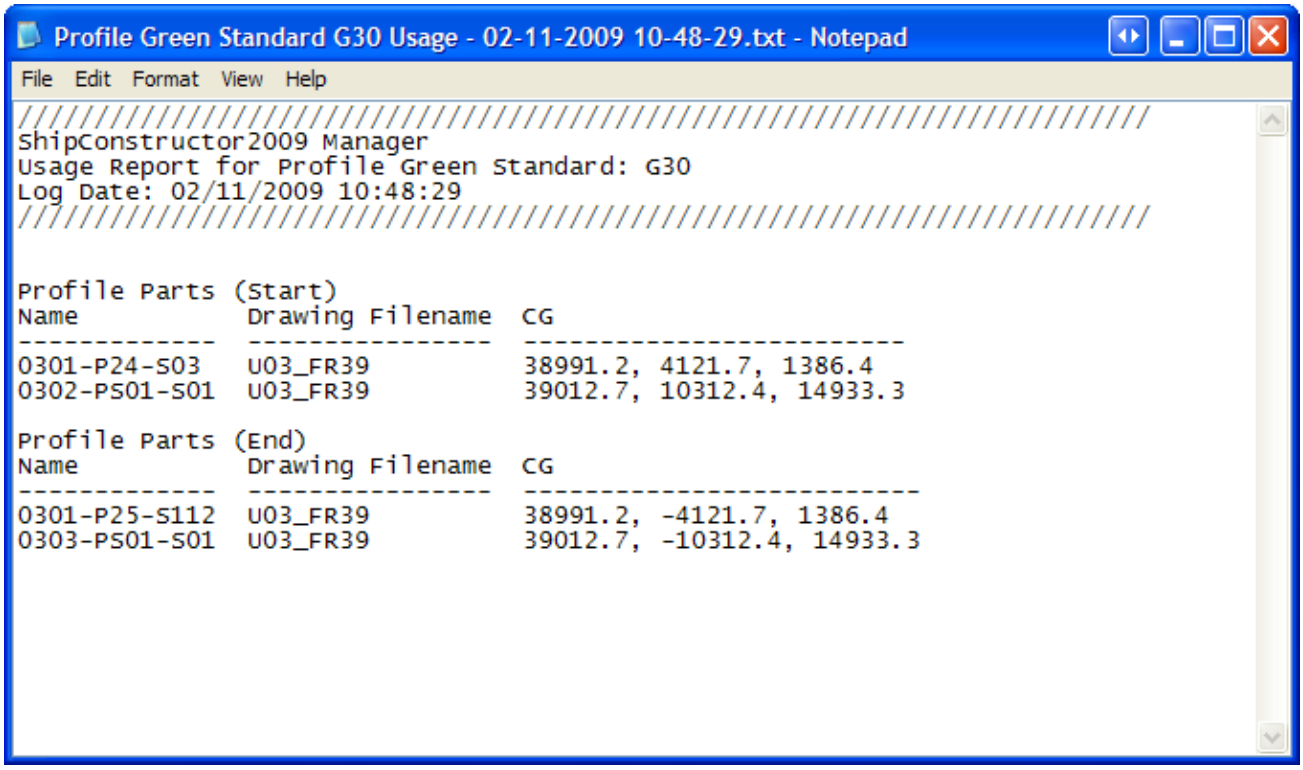
View Profile Green Standard Usage

It is possible to see a list of all the parts that use a selected profile green standard. A check mark in the In Use column indicates that the profile green standard is being used by parts.

To list all parts using a specific profile green standard

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Profiles > Green Standards (page 361) to open the Profile Green Standards window.
3. Select the profile green standard you want.
4. Click Usage Log.

   ShipConstructor displays a list of all parts using the selected profile green standard.


Profile Green Standard G30 Usage - 02-11-2009 10-48-29.txt - Notepad
File Edit Format View Help
//
ShipConstructor2009 Manager
Usage Report for Profile Green Standard: G30
Log Date: 02/11/2009 10:48:29
//
Profile Parts (Start)
Name Drawing Filename CG
0301-P24-S03 U03_FR39 38991.2, 4121.7, 1386.4
0302-PS01-S01 U03_FR39 39012.7, 10312.4, 14933.3
Profile Parts (End)
Name Drawing Filename CG
0301-P25-S112 U03_FR39 38991.2, -4121.7, 1386.4
0303-PS01-S01 U03_FR39 39012.7, -10312.4, 14933.3

Standard Parts Definition

Standard parts are usually brackets or similar items. By setting them up as standards, you can simply insert them into your design during modeling. ShipConstructor lists them separately in BOMs. Often standard brackets are NC cut using fixed routines at the NC-cutting machine. (For example, standard brackets can be cut using multiple torches, reducing NC-cutting time.)

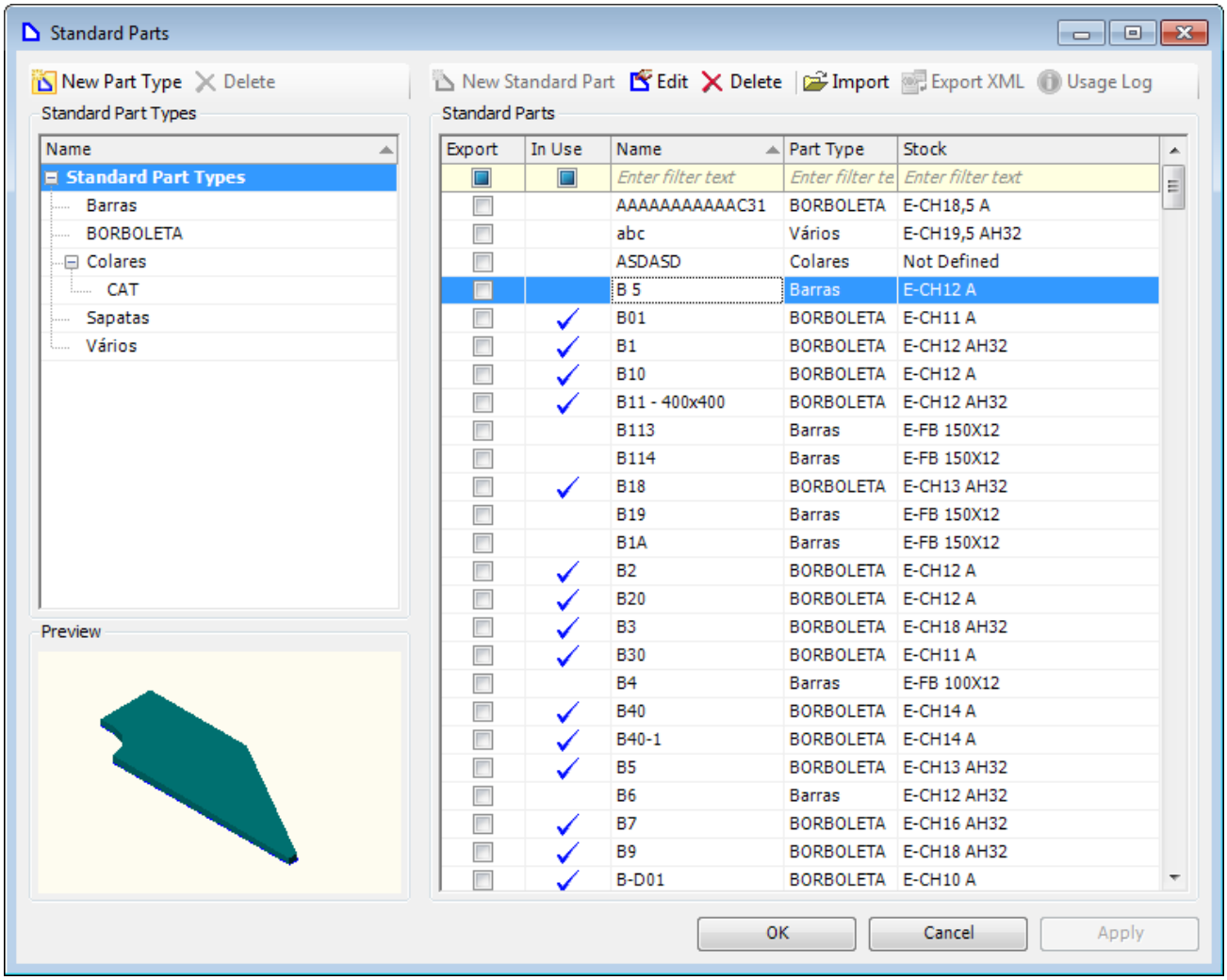
Standard parts are plate or stiffener parts that are shipyard standards. Standard parts can be arranged by types in the Standards Part Types tree. Standard parts can be changed after parts are inserted into the structure model.

Set Up Standard Part Types

You can organize standard parts by setting up standard part types and organizing them into groups and sub-groups.

To set up standard part types

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Standard Parts (page 362) to open the Standard Parts window.



3. Under Standard Part Types, select the group that you want to create a sub-group within. By default there is one group: Standard Part Types.
4. Click New Part Type to create a new standard part type.
5. Enter a name for the group.
6. Continue creating groups and sub-groups.
7. Click Apply to save changes or OK to save changes and close the Standard Parts window.

Create a Standard Part

To create a standard part

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Standard Parts (page 362) to open the Standard Parts window.
3. Under Standard Part Types, select the group or sub-group that you want to create a standard part within.
4. Click New Standard Part to create a new standard part.
5. Enter a name for the part and press Enter.

Edit a Standard Part

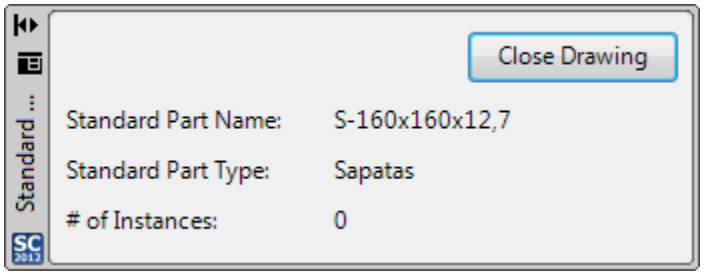
To edit a standard part

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Standard Parts (page 362) to open the Standard Parts window.
3. Under Standard Part Types, select the group or sub-group that the standard part you wish to edit belongs to.
4. Under Standard Parts, select the part you want to edit.
5. Click Edit to switch to AutoCAD.

   Note: You will be asked to save if needed before continuing to edit.

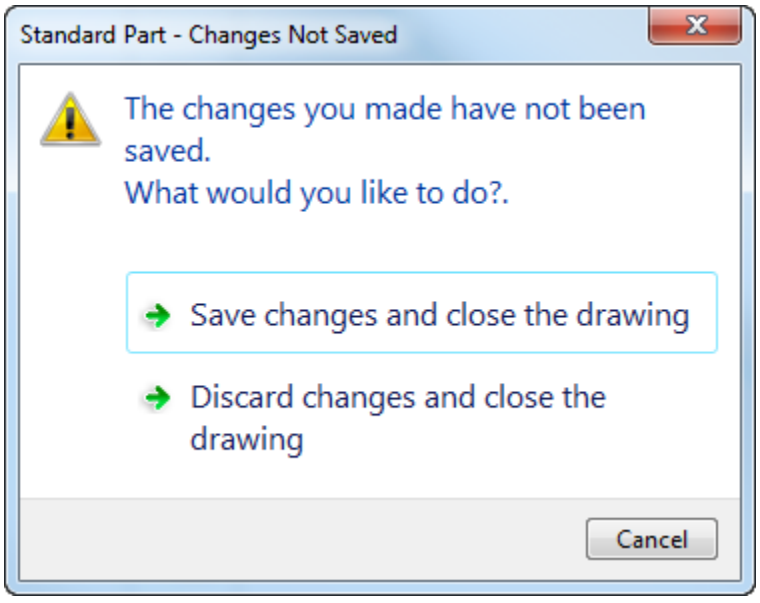
6. Create a plate part as you would normally do in the structure model. The (0,0) position in the drawing determines the insertion point of the standard part.

   The Standard Part Palette appears when editing the standard part in AutoCAD. This palette can be docked or rolled up like any other AutoCAD palette.



7. Click Close Drawing in the palette to return to Manager.

   You will be asked to save if changes were made to the drawing. Alternatively, use AutoCAD save.


Standard Part - Changes Not Saved
The changes you made have not been saved.
What would you like to do?.
Save changes and close the drawing
Discard changes and close the drawing
Cancel

   Note: There must be a stock defined in the Structure Stock Catalog of the type of standard part you want to create

   All other rules of creating plate parts apply when creating a standard part.

Rename a Standard Part

To rename a standard part

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Standard Parts (page 362) to open the Standard Parts window.
3. Under Standard Part Types, select the group or sub-group that the standard part belongs to.
4. Under Standard Parts, double-click the Name field and enter a new name.
5. Click Apply to save changes or OK to save changes and close the Standard Parts window.

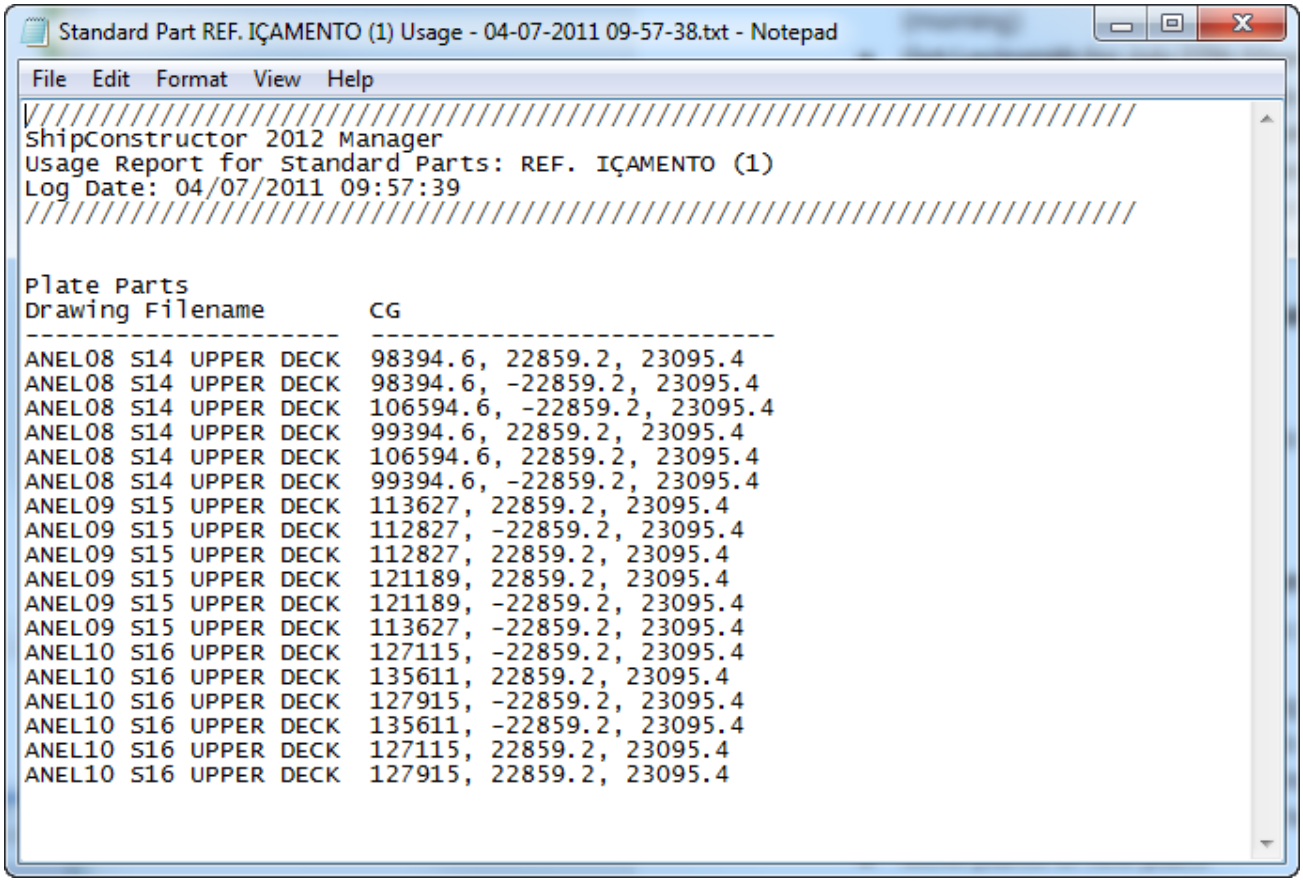
View Standard Part Usage

It is possible to see a list of all the parts of a selected standard part. A check mark in the In Use column indicates that the standard part exists.

To list all parts of a specific standard part

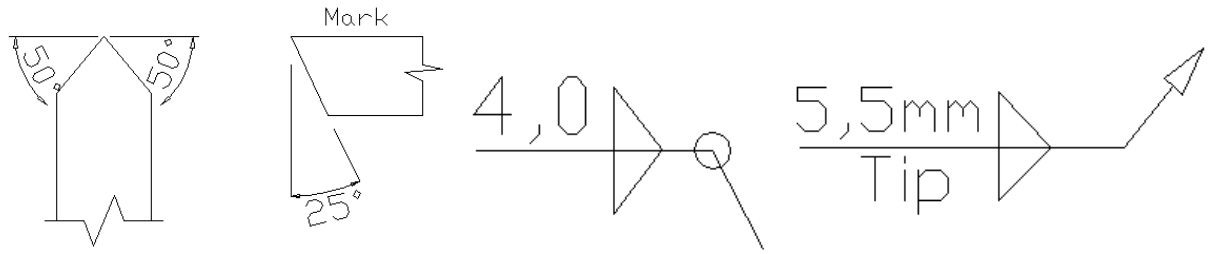
1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Structure > Standard Parts (page 362) to open the Standard Parts window.
3. Select the standard part you want.
4. Click Usage Log.

   ShipConstructor displays a list of all parts of the selected standard part.



Dynamic Marking Block Standards

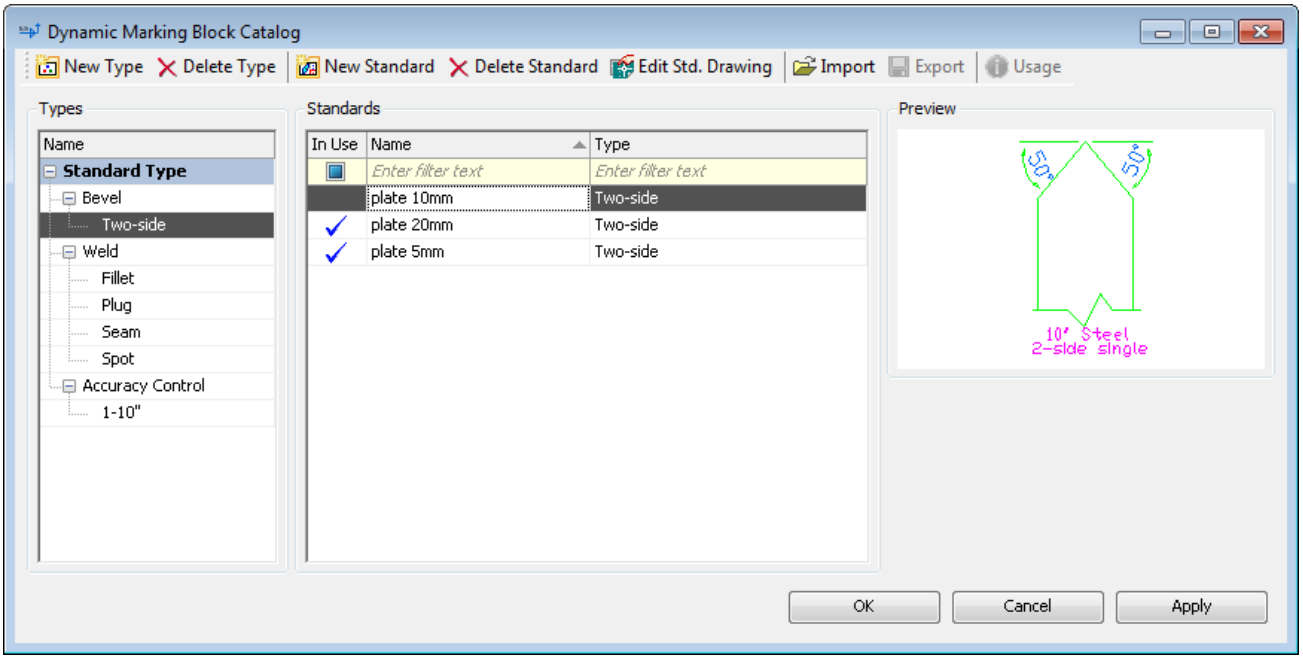
A dynamic marking block (DMB) is a grouping of polylines and text with dynamic elements which can be placed and positioned on a structure part as if it was a single entity. This makes it much easier to edit, update, and maintain the markings on multiple parts. The following are several examples of DMBs:



Dynamic marking block standards are created through the Dynamic Marking Block Catalog.

To open the dynamic marking block catalog:

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Dynamic Marking Blocks (page 362) to open to the Dynamic Marking Block catalog. The DMB catalog is shown below.



Set Up Dynamic Marking Block Types

Types provide a way for users to organize dynamic marking block standards into a folder like structure. Use the New Type and Delete Type buttons to create and delete types. To rename a type, edit the name directly in the type window. You will need to have at least one type to be able to create DMB standards.

Note: You cannot delete a type if a DMB standard is assigned to that type or any of any of its sub types.

Create a Dynamic Marking Block Standard

Once there is at least one DMB type, a dynamic marking block standard can be created. Select the DMB standard type where the new standard should be placed and use the New Standard button to create a new DMB standard. Edit the name of the standard directly in the standards table.

Delete a Dynamic Marking Block Standard

To delete a dynamic marking block standard use the Delete Standard button. A standard cannot be deleted if there are instances of the DMB in the project.

Edit a Dynamic Marking Block Standard

To create the actual geometry and text in the dynamic marking block standard, or edit and existing standard, select it in the Standards table and click the Edit Std. Drawing button. This will prompt you to save any changes you have made in the Dynamic Marking Block Catalog and open up a drawing that contains the DMB standard entities. In this drawing, users can create\edit\delete geometry and text that will be part of the dynamic marking block.

Note: If you have multiple dynamic marking block standard drawing templates you will be prompted to select which template to use for the DMB standard the first time it is edited.

After the the Standard Assembly Standard drawing opens, there will be a DMB palette that contains some basic information about the DMB and its type. The Close Drawing button will close the drawing and return to manager with the option to save or discard changes made to the DMB standard.

Note: This is an AutoCAD palette and can be docked or rolled up in AutoCAD.

Use the standard AutoCAD drawing tools to create the dynamic marking block. Most AutoCAD entities are valid in the DMB standard. Any invalid objects are removed when the drawing is saved.

LAYERS

A dynamic marking block standard drawing has two default layers: _Mark and _NoProcess. Other layers may be created by the user but these will not be stored in the database and will not be present if the standard drawing is ever deleted and re-created. Entities on custom layers will be treated the same as entities on the _Mark layer.

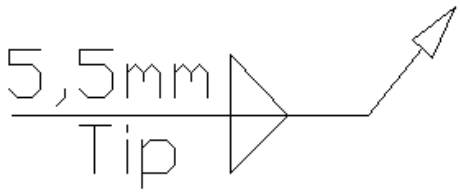
GEOMETRY

Most 2D AutoCAD geometry is valid when editing the DMB standard. Any entity that can be converted to a light-weight polyline without loss of information should be accepted (line, circle, arc, etc.). More complex objects, such as NURBS curves and AcDb3Dpolylines are not valid. When the drawing is saved all valid objects are converted to simple polylines. Any invalid geometry is removed from the drawing.

TEXT

Use the AutoCAD MTEXT or TEXT commands to add text to the DMB standard. Text in a DMB standard can be static or dynamic. Static text cannot be overridden on individual instances of the dynamic marking block. Dynamic text can contain different values on different instances.

For example, in the following DMB, the text “Tip” and “mm” may be static, as they would not change in each instance of the DMB, but the “5,5” may be dynamic to allow this dynamic marking block to be used with different values.



All text is dynamic by default. To set text static, use the command Set Dynamic Marking Block Text Static (page 471) or right-click the text in the drawing and check “Static” in the menu. To set text dynamic, use the Set Dynamic Marking Block Text Normal (page 471) command, or right-click the text and un-check “Static” in the menu. When the DMB is added to a structure part, there will be an option to override dynamic text.

Note: when text in the standard is changed from dynamic to static, any text that has been overridden in individual instances of the standard will revert back to whatever the static text is in the standard.

When an instance of the DMB is added to a part, the origin of this standard drawing will be positioned at the cursor. This is the insertion point used to position the DMB in the model.

When finished editing the standard, use the Close Drawing button on the DMB palette to save or discard changes and return to manager.

Import Dynamic Marking Block Standards

Dynamic marking block standards can be imported from another project. To import DMB standards, use the Import button. Browse to the project file (*.pro) of a project that contains DMB standards. Select the types and standards to be imported and click OK. Any standard that does not exist in the current project will be imported.

Export Dynamic Marking Block Standards

Export functionality is not currently available for dynamic marking block standards. To use the current DMB standards in another project, open the dynamic marking block catalog in the other project and import the DMB standards from this project.

View Dynamic Marking Block Standard Usage

If the currently selected dynamic marking block standard is in use in the current project the Usage button will be enabled. Click this button to display each part and drawing that uses the current DMB.

Preview Dynamic Marking Block Standard

A preview of the selected dynamic marking block standard is shown in the catalog. This preview shows a snapshot of the DMB standard drawing the last time it was saved.

Drawing Templates

ShipConstructor creates drawings that let you specify templates to propagate standards. These templates are created on the Templates page of Navigator. Template drawings can contain styles, user layers, views, and possibly objects. Changes to template drawings do not affect existing drawings.

Required Templates

- Assembly
- PipeSpool
- HVACSpool
- ProfilePlotSheet
- ProfilePlot
- Nest
- PipeArrangement
- HVACArrangement
- EquipmentArrangement

Optional Templates

- Structure
- Nest Drawing
- Pipe
- HVAC
- Equipment
- Hull
- PinJig
- Approval
- ProfilePlotDrawing
- Composite

Optional templates only allow one template drawing. If no template drawing is available, a blank AutoCAD drawing is created. It is not the acad.dwt template.

To create a new template from an existing non-ShipConstructor drawing

1. If the template you are creating is an optional template, remove any existing templates from Navigator.
2. In Windows Explorer, copy the drawing to the correct folder in ShipConstructor.
3. In Navigator, select the Templates page.
4. Open the drawing that was copied into ShipConstructor.
5. After the drawing is open you will be asked to register the drawing. Click Yes.

To delete a template drawing

1. In Navigator, select the Templates page.
2. Select the template drawing you want to delete.
3. Right-click and choose Delete.
4. Click Yes to confirm the delete.

To rename a template drawing

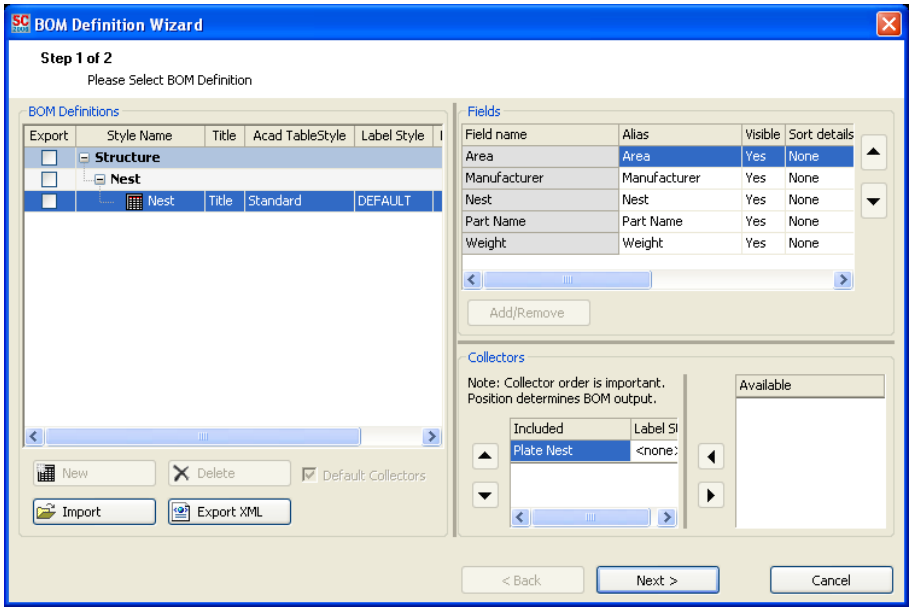
If you renamed a template drawing using Windows Explorer, Navigator will display two icons: a gray one with the old filename, and a blue one with the new filename. Choosing to open either drawing will prompt you to update the database with the new filename.

1. In Navigator, select the Templates page.
2. Select the template drawing you want to rename
3. Right-click and choose Rename.
4. Enter the new name.

Insert Empty BOM Wizard

The BOM Definition Wizard is used for inserting instances of a particular BOM into a drawing. Multiple tables can be inserted into a single drawing.

1. Step 1 of the wizard lets you specify which BOM definition you want to insert in the BOM. The interface is similar to the BOM Definitions window; however, it is a read-only view for selection purposes. Select a BOM definition to advance to the next page.



2. Step 2 lets you select BOM types. BOM functionality depends on the production drawing type. All drawing types except for Nest and Profile plots require that the BOM be inserted in paper space only. Nest and Profile Plots are inserted in model space only.

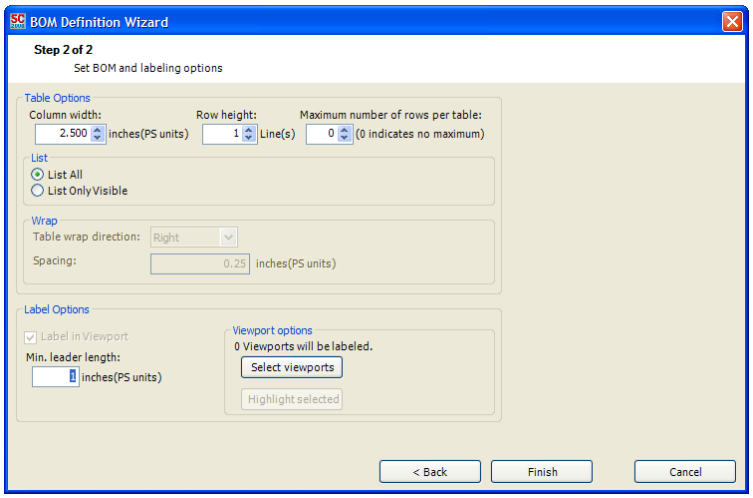


Table Options

All rows generated by the collectors are compiled into the BOM data master set. You can control the items listed in the BOM by choosing one of the two options:

- List All – Lists all items in the compiled master BOM.
- List Only Visible – Items displayed in the BOM will be based on objects' viewport visibility. Items that are visible in any of the viewports within the layout will be listed. Object layer visibility and viewport clipping are respected.

The numbering of items remains consistent across all layouts regardless of whether List All or List Only Visible is chosen.

Maximum number of rows per table – Specifies the maximum number of data rows that a table will contain before a new table is created.

Wrap Properties – Choices are enabled when a value greater than zero is entered in the Maximum number of rows per table field.

- Table wrap direction – Direction that new tables should be added.
- Spacing – Space between newly created tables.

Label Options

The labeling procedure works in conjunction with the BOM. Only objects that are referenced by a particular BOM are labeled. Label text is retrieved from the column specified by the BOM definition.

The label style used for labeling is defined in the BOM definition.

ShipConstructor marks viewports that you select with AutoCAD xdata. Once selected, the objects in the viewport will be labeled every time the SCRELABELALL or SCRELABEL command is run. You can manage selected viewports in two ways:

  - Using the BOM Definitions Wizard
  - Using the SCVPORTOPTIONS command

- Min leader length -The minimum leader length used when labeling. This value is stored on a per-viewport basis. The wizard remembers the last used value per layout.

Structure Setup

Import a Hull Model

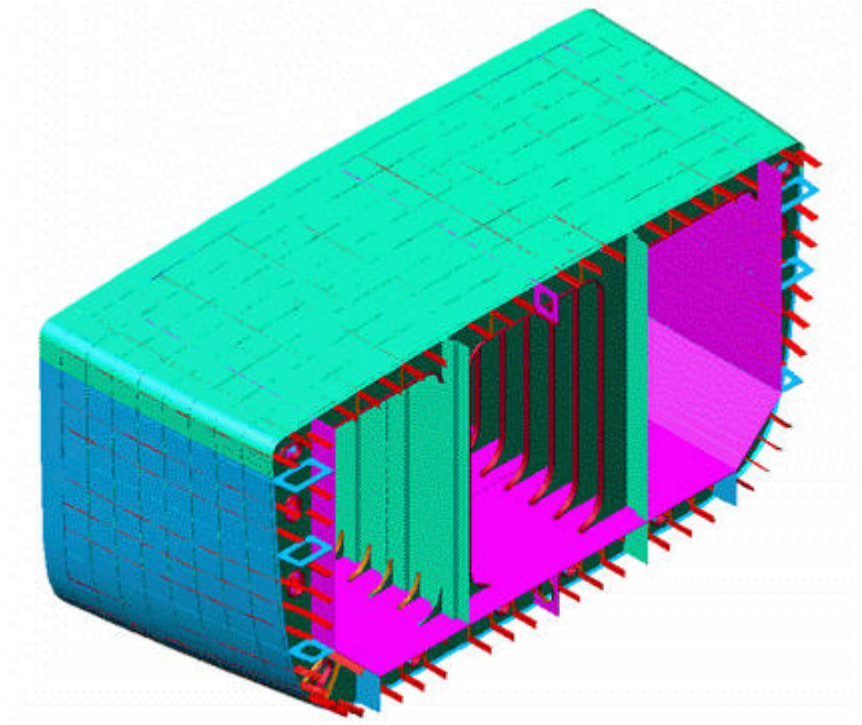
You can import a 3D wireframe model (frame lines, longitudinal bulkhead lines, deck lines, and so on) from Hull, ShipCAM, or another fairing program as a starting point for structural modeling. If you are using the Hull module for lofting operations (plate definition and expansion), the model needs to be inserted into a Hull drawing. Otherwise, proceed to creating units before importing data.

Use a Hull Model from the Hull Module

See the Hull manual.

Unit Setup

You must decide how you want to divide your vessel (project) into production units (also known as blocks). The size of a unit is usually determined by the maximum weight that the shipyard can handle. Once you have decided where the unit breaks will occur, you can set up and name the units in ShipConstructor. When you create a unit in ShipConstructor, you simultaneously create a unit drawing.



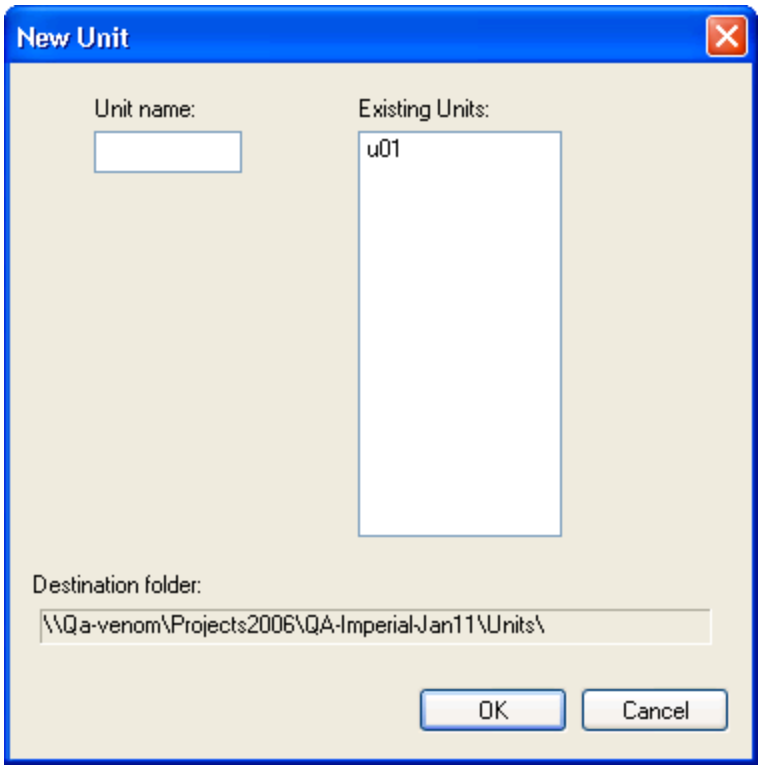
A typical unit

Create Units

To create a unit

Note: You can create a unit from within an empty drawing or from within a drawing that contains the sections for the new unit.

1. In the Navigator, select the Project page.
2. Click New Unit to open the New Unit window.



3. In the Unit Name text box, type a unit name.
4. Select whether to create a blank drawing as a new unit drawing (the default), or use the current drawing as a new unit drawing.
5. Click OK.

Delete Units

There is no easy way to delete a unit, because each unit is linked to so many other parts of a project.

To delete a unit

1. Use the Navigator to delete all planar group drawings, pipe drawings, HVAC drawings, and equipment arrangement drawings for the unit.
2. Use Windows Explorer to delete the folder for the unit.
3. Use the Product Hierarchy window to delete the unit.

Product Hierarchies

Product hierarchies are the sequence of grouping parts together. The build strategy is the main product hierarchy. The build strategy is the sequence that the parts in your project will be physically assembled in. Product hierarchy is also referred to as product work breakdown structure (PWBS).

Product hierarchies consist of several levels. At the lowest level, individual parts are assembled to form minor assemblies. At the next level, minor assemblies are combined to form panels. At subsequent levels, panels are combined to form assemblies, assemblies are combined to form stages, stages are combined to form units, and units are combined to form the completed project.

The first step in developing a product hierarchy is to divide your project into units. See Unit Setup (page 107). The top level of the product hierarchy is always the project, and the project is always divided into units. Typically you perform this step of the product hierarchy before you begin designing parts. (Some shipyards prefer to name their project based on the unit they belong to.)

Next, you set up the available sub-levels that you potentially want to use within your units. See Set Up Assembly Levels (page 116). These levels are referred to as assembly levels. The levels listed above are ShipConstructor's default assembly levels, but you can modify them as necessary.

Note: The word assembly has several meanings. In the general sense, an assembly (or assembly level) refers to any level within or under the unit. In the specific sense, an assembly refers to the level above panel (that is, an assembly is composed of several panels). An assembly (or assembly item) can also mean any item of several parts (for example, assembly XYZ could refer to a panel, a unit, or other item).

Next, you decide which specific stages to use for each specific unit, which specific assemblies to use for each specific stage, and so on. See Set Up Assemblies (page 119). For example, you may want to divide unit U12 into five stages: COLDOUTFIT, HOTOUTFIT, HVAC, PIPING, and STRUCTURE. At the lowest level, you assign individual parts to assemblies. See Assign Parts, Spools, or Assemblies to Assemblies (page 122). Obviously, you cannot perform this step in the product hierarchy process until you have actually created your parts.

When assigning parts to assemblies, it is helpful to use a product hierarchy drawing so you can see all parts within a unit. See Setting up Multiple Product Hierarchies (page 109).

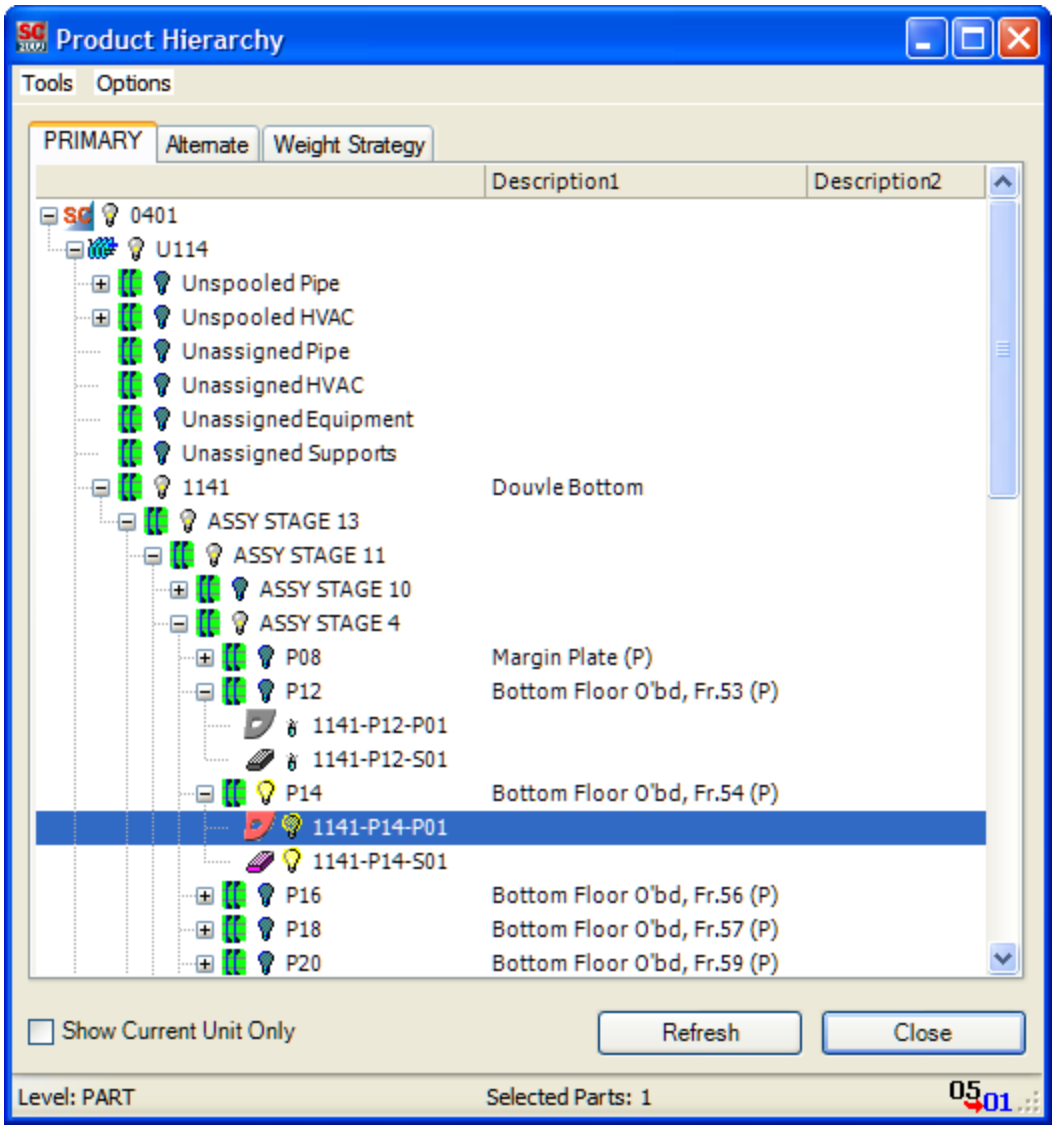
Finally, you must decide on a time schedule for each assembly, indicating when it will be completed.

Setting Up Multiple Product Hierarchies

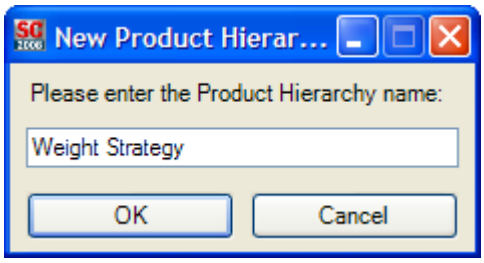
You can set up multiple hierarchies within a project. For example, in addition to the build strategy used for assembly, you may want to create a separate hierarchy used for a SWBS breakdown, another for weight quality control and analysis, and a third for paint planning. There is a designated product hierarchy (called the build strategy) that is the only product hierarchy used for nesting, creating assembly drawings, and creating arrangement drawings.

To create a new product hierarchy

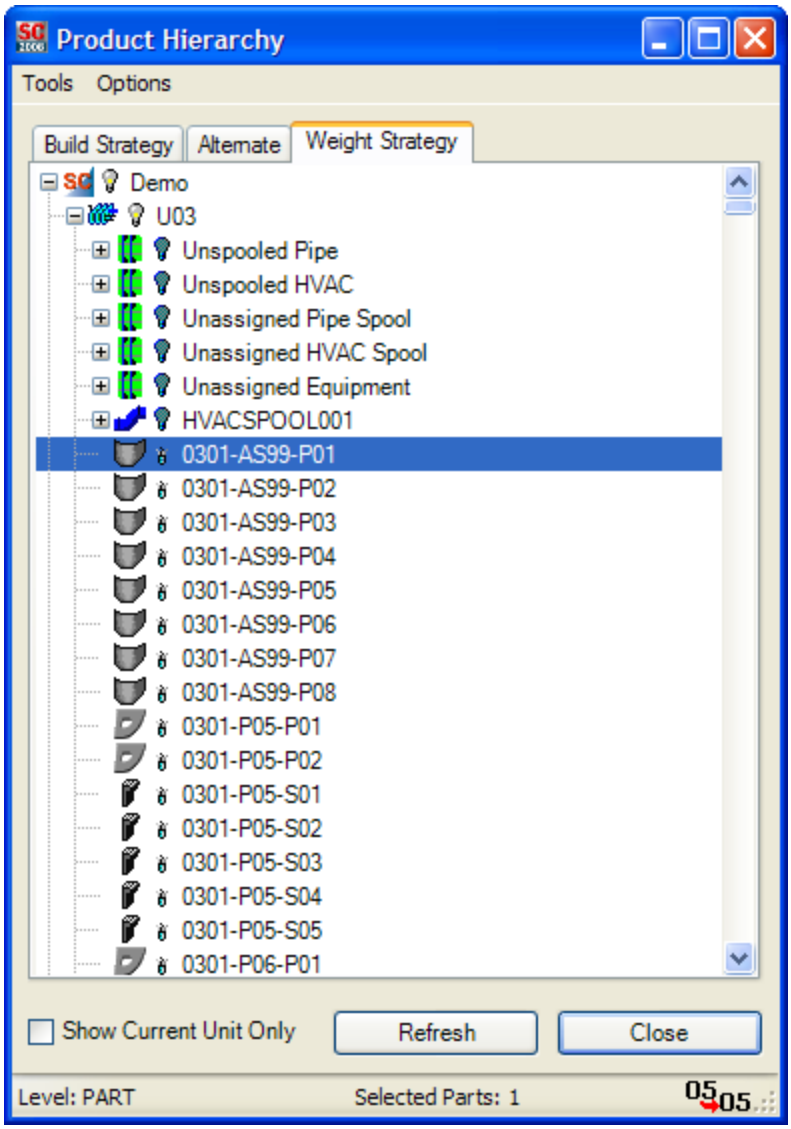
1. Click in the Assembly Toolbar.



2. Choose Tools > Product Hierarchy > New.



3. Enter a new Product Hierarchy name and click OK.



All parts will be assigned to the appropriate unit assembly of the new product hierarchy. Each product hierarchy has its own levels.

Copy Product Hierarchy

This feature allows you to create a new Product Hierarchy from an existing Product Hierarchy. You can choose if you want to copy Assembly and Part hierarchy, or just the Assembly hierarchy.

What is copied:

- Assemblies
- Assembly Levels
- Assembly User-Defined Attributes
- Finishes
- Part locations if you choose to copy Assembly and Part hierarchy.

Notes: If you are copying the Primary Product Hierarchy, Units and Standard Assemblies will become Assemblies in the new Product Hierarchy.

If you choose to copy only the Assembly hierarchy, parts will be located in their default “Unassigned” Assembly. For example, Structure parts would be located in the “Unassigned Structure” Assembly.

To copy an existing product hierarchy

1. Open the Product Hierarchy dialog.
2. Select the Product Hierarchy you wish to copy.
3. From the menu, select Tools > Product Hierarchy> Copy...
4. Provide the name of the new Product Hierarchy
5. If you want to copy part assignments to Assemblies, select Copy Part Locations
6. Click OK

To copy an existing product hierarchy from the command line

1. Run the -SCProductHierarchyCopy command.
2. Provide the name of the Product Hierarchy that you want to copy.
3. Provide the name of the new Product Hierarchy.
4. Indicate if you want to copy the Assembly and Parts hierarchy (yes) or just the Assembly hierarchy (no).

A new Product Hierarchy will be created, and can be viewed in the Product Hierarchy dialog.

Export a Product Hierarchy to another ShipConstructor project

A product hierarchy can be exported to an xml file for importing into another ShipConstructor project.

To export an existing product hierarchy

1. Open the Product Hierarchy dialog.
2. Select the Product Hierarchy you wish to export.
3. From the menu, select Tools > Product Hierarchy> Export...
4. Provide the name of the xml file for saving the export data.

To export an existing product hierarchy from the command line

1. Run the -SCProductHierarchyExport command.
2. Provide the name of the Product Hierarchy that you want to export.
3. Provide the name of the xml file for saving the export data.

Import a Product Hierarchy from another ShipConstructor project

A product hierarchy can be imported from an xml file that was exported from another ShipConstructor project.

What is imported:

- Assemblies
- Assembly Levels
- Assembly User-Defined Attributes

Note:If you are importing a Primary Product Hierarchy, Units and Standard Assemblies will become Assemblies in the new Product Hierarchy.

To import a product hierarchy

1. Open the Product Hierarchy dialog.
2. From the menu, select Tools > Product Hierarchy> Import...
3. Provide the name of the xml file containing the importing Product Hierarchy.

   If the import is successful, a new Product Hierarchy will be created.

To export an existing product hierarchy from the command line

1. Run the -SCProductHierarchyImport command.
2. Provide the name of the xml file containing the importing Product Hierarchy.

   If the import is successful, a new Product Hierarchy will be created. To view this new product hierarchy, open the Product Hierarchy dialog.

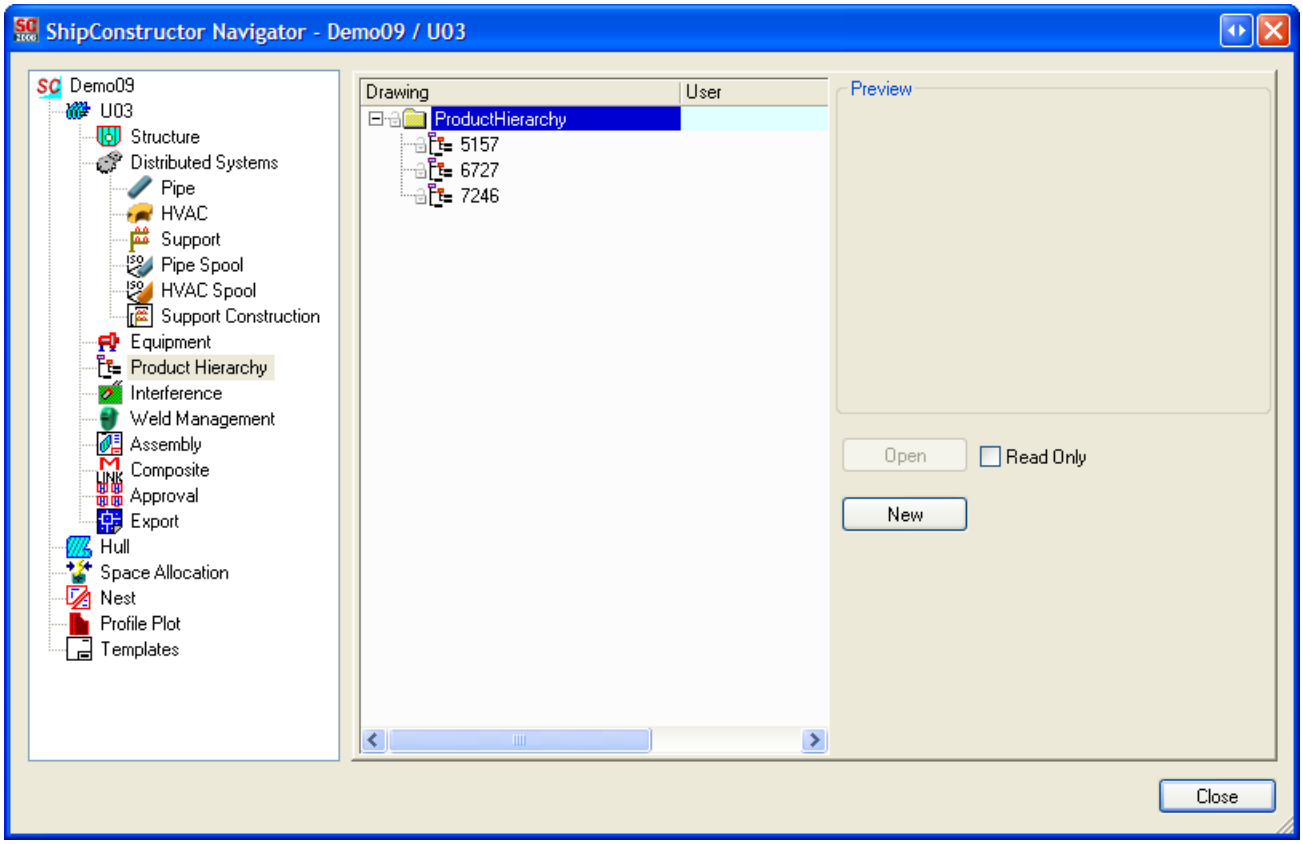
Set Up Product Hierarchy Drawings

A product hierarchy drawing contains all the parts for a unit, including structure, equipment, piping, HVAC, and so on. You usually set up a product hierarchy drawing for a unit so you can see the parts as you develop a product hierarchy. You can make a specific assembly visible and check that all required parts are in that build stage or change the parts involved in a build stage by dragging them to another build stage.

Create a Product Hierarchy Drawing

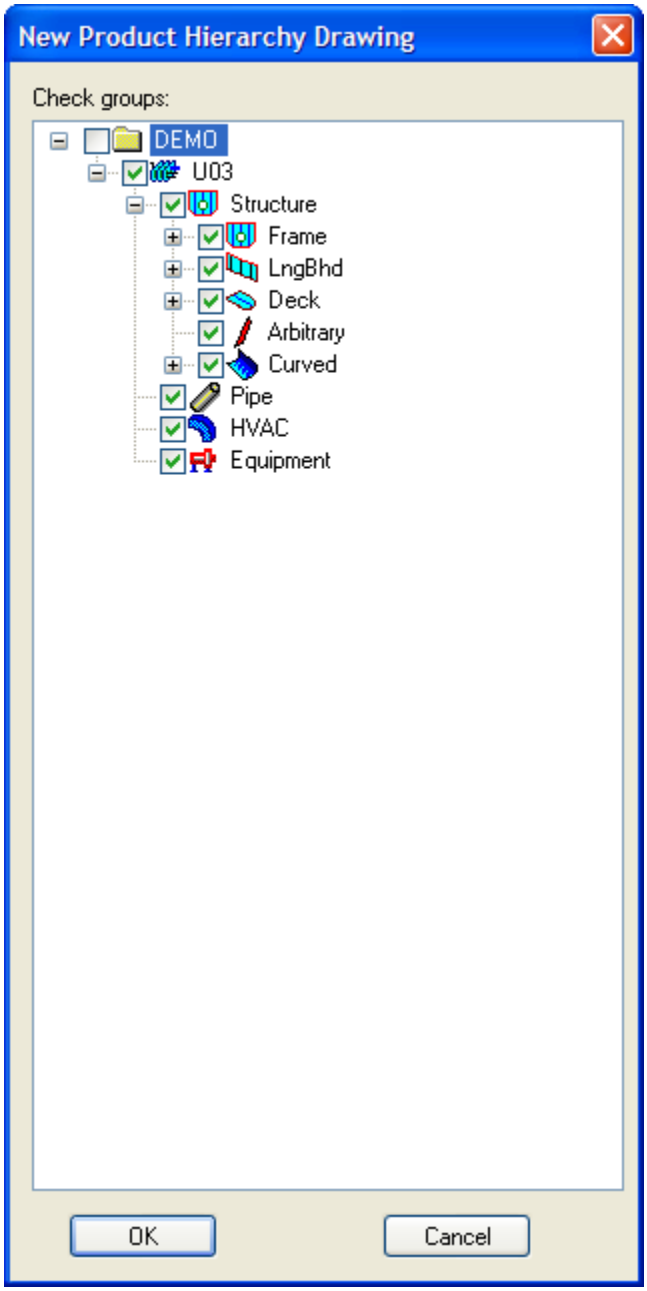
To create a product hierarchy drawing

1. In Navigator, select the Product Hierarchy page.



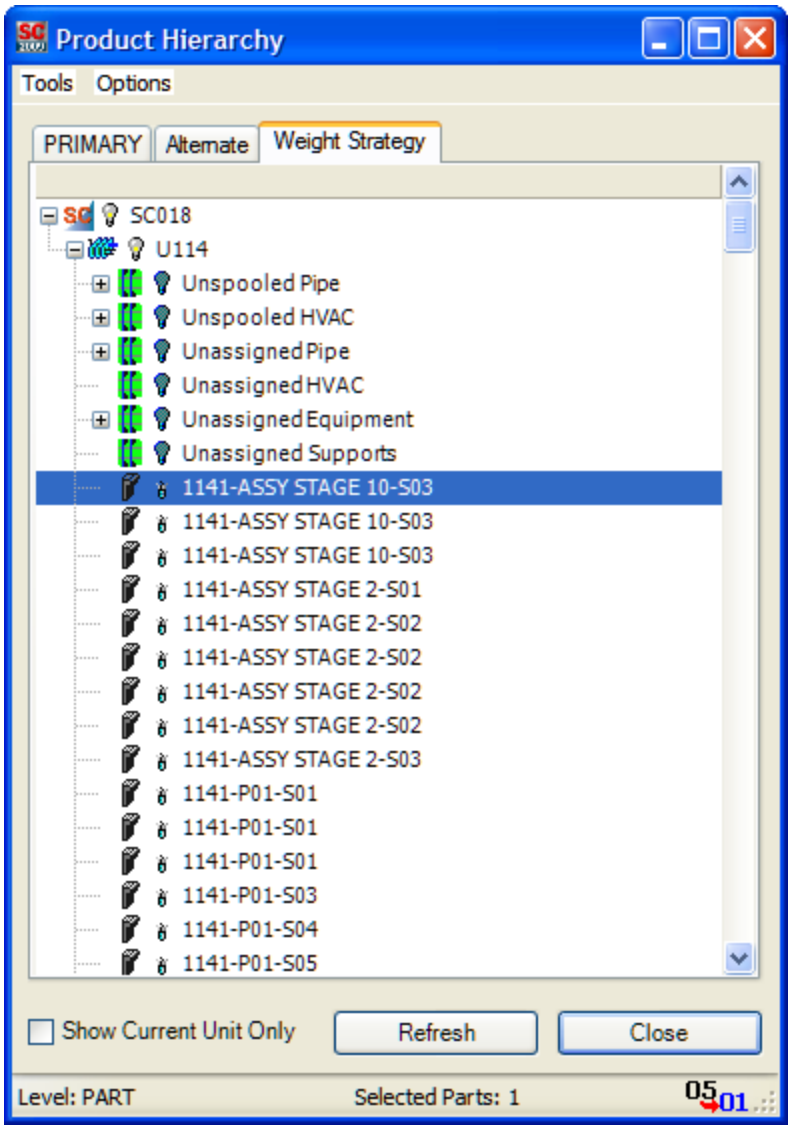
2. Click New.

   The New Product Hierarchy Drawing window appears.



3. Select the drawings to include in the product hierarchy drawing.
4. Click OK.

   The New Drawing window appears.



5. Enter a name for the drawing.
6. Click OK.

> Note: If any parts are created after you create a product hierarchy drawing, the new parts will not appear in the drawing unless you update it.

Rename a Product Hierarchy Drawing

To rename a product hierarchy drawing

1. In Navigator, select the Product Hierarchy page.
2. Right-click the product hierarchy drawing and choose Rename.
3. Enter a new name and press Enter.

Delete a Product Hierarchy Drawing

To delete a product hierarchy drawing

1. In Navigator, select the Product Hierarchy page.

2. Right-click the product hierarchy drawing and choose Delete.

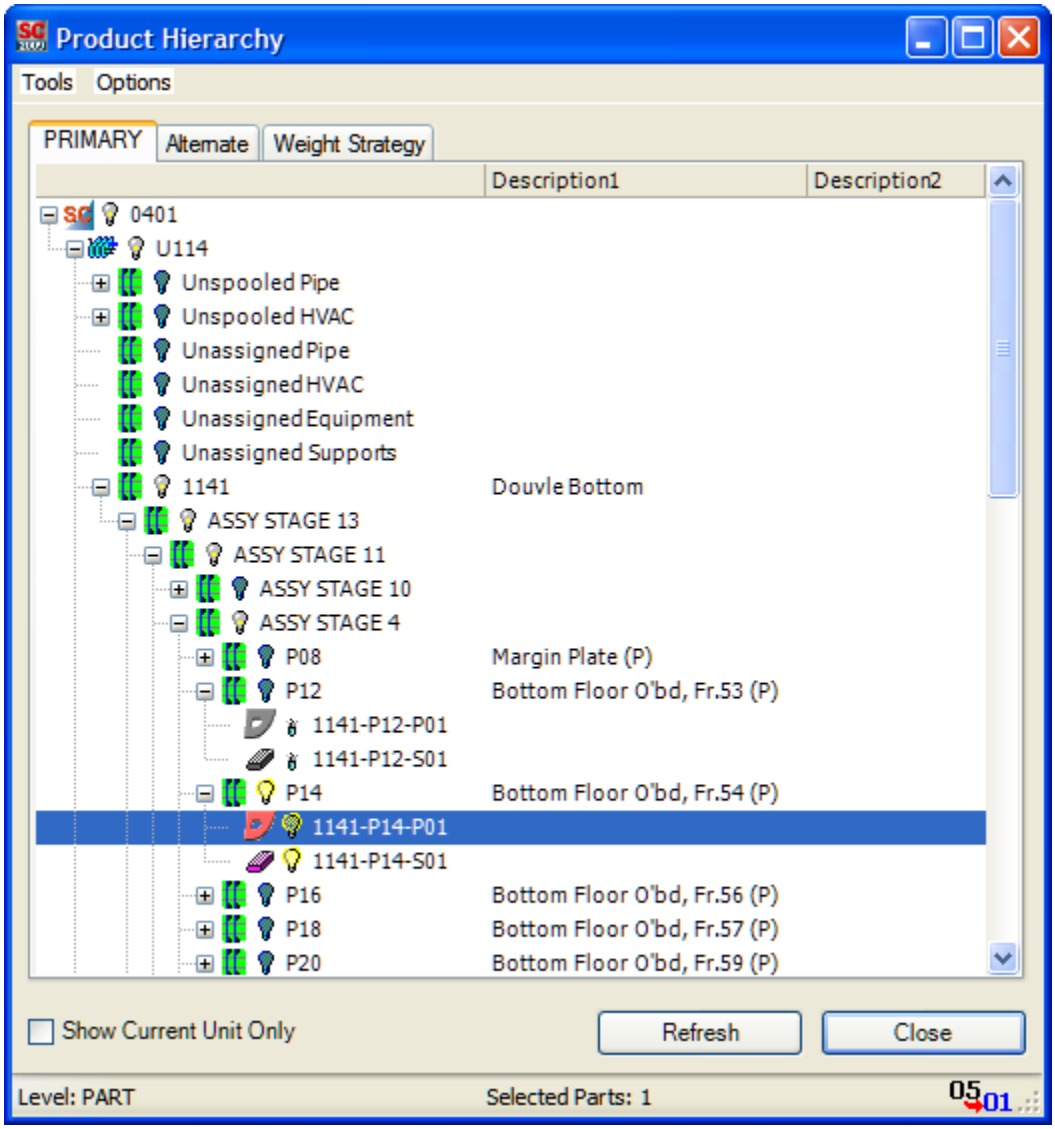
Set Up Assembly Levels

To set up assembly levels

Note: You may want to create a product hierarchy drawing before setting up assembly levels (see Create a Product Hierarchy Drawing (page 113)). The product hierarchy drawing will help you see the parts as you develop a product hierarchy. You can make a specific assembly visible and check that all required parts are in that build stage or change the parts involved in a build stage by dragging them to another build stage. (If you open an old product hierarchy drawing, any parts that have been created after the drawing was created or updated do not appear in the drawing.)

1. Do either of the following:
   - Within a product hierarchy drawing, choose SC Product Hierarchy > Develop Product Hierarchy (page 508)
   - Within any drawing, click on the Assembly Toolbar (page 477)

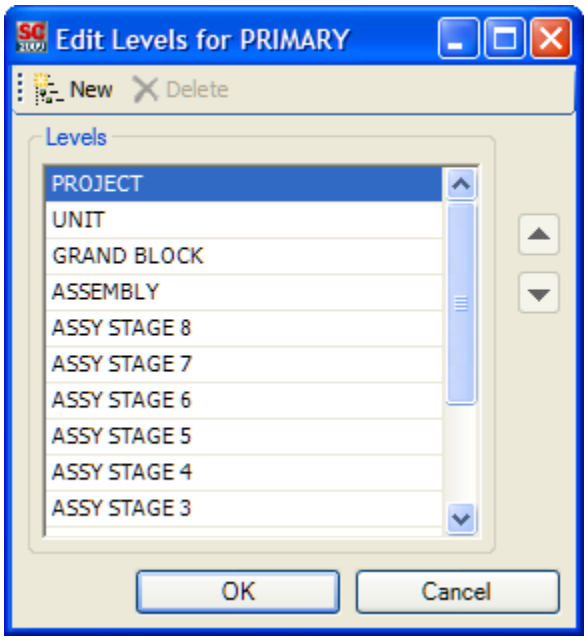
   The Product Hierarchy window appears.



Note: When the Product Hierarchy window is open, you can continue to perform commands in the drawing, such as changing your view, changing the shading mode, and so on.

The tabs represent different product hierarchies. The default product hierarchy is Build Strategy. To select a different product hierarchy, click its tab. To create a new product hierarchy, choose Tools > Product Hierarchy > New and enter a name for the product hierarchy. To delete a product hierarchy, click its tab and then choose Tools > Product Hierarchy > Delete. To rename a product hierarchy, click its tab and then choose Tools > Product Hierarchy > Rename.

2. Choose Tools > Edit Levels to open the Edit Levels window.



This window lists the levels of the current product hierarchy. Later, you will assign parts and assemblies to the various levels. A typical configuration for levels is:

- Project – The top level of the product hierarchy.
- Unit – Every project must have at least one unit. A typical project consists of several units. A smaller project may have only one unit (Hull plus wheel house).
- Stages – Often the final assembly is carried out using many stages that usually combine assemblies and panels. A unit can contain several levels of stages; in some cases, there can be twenty levels of stages between the assembly and the unit.
- Assembly – An assembly (also known as a 3D assembly) consists of several panels, plus some individual parts. A typical assembly might be a double bottom, a wing tank, or something similar.
- Panel – A panel may be flat or curved and traditionally consists of one or more plate parts plus several stiffeners.

You can add, delete, or re-order levels to suit your project.

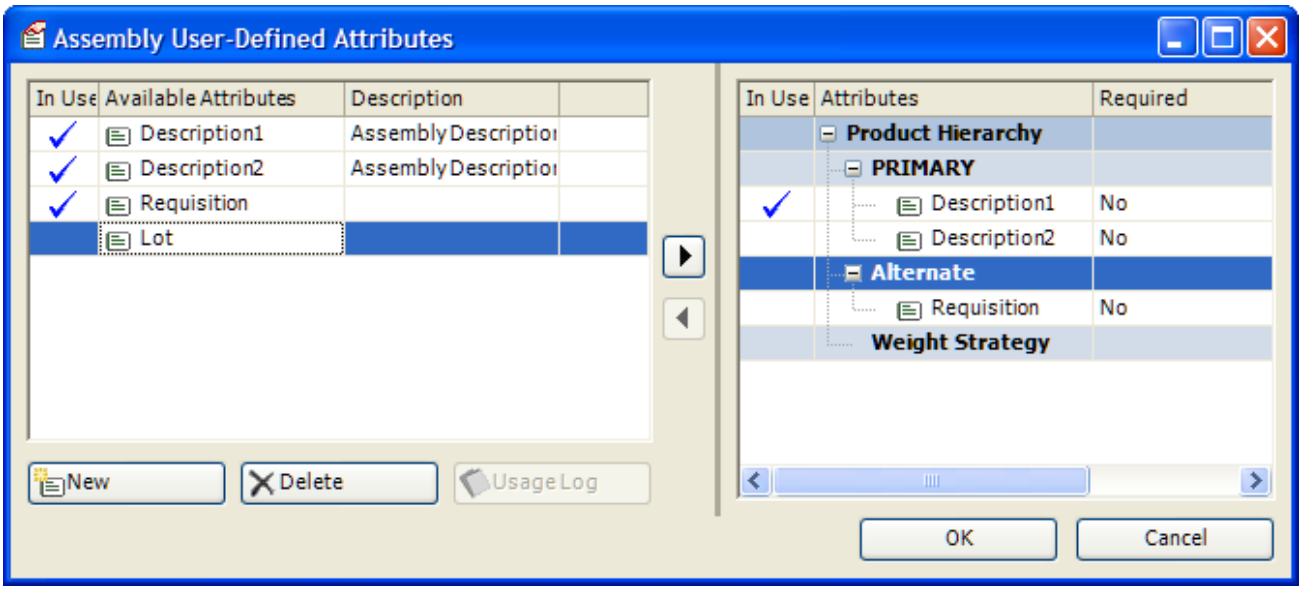
To add a level, select the level to create a new level under, click New, and enter a name for the level.

To delete a level, select the level and click Delete.

To re-order a level, select the level and click the arrow buttons to move the level up or down.

Set Up User-Defined Attributes

You can create your own attributes for assigning values to Assemblies. User-defined attributes are assigned to a product hierarchy, and each Assembly in that product hierarchy will have its own attribute values. For more information, see User-Defined Attributes (page 49).



To create a new user-defined attribute

1. From the Product Hierarchy window (see Set Up Assembly Levels (page 116)) menu, select **Tools > User-Defined Attributes...**
2. Click **New**.
3. Enter a name for the attribute and press **Enter**.
4. Optionally, add a description.
5. Click **OK** to close the **User-Defined Attributes** window.

To assign an user-defined attribute to a Product Hierarchy

1. From the Product Hierarchy window (see Set Up Assembly Levels (page 116)) menu, select **Tools > User-Defined Attributes...**
2. Select the attribute on the left side of the window.
3. Select a product hierarchy on the right side of the window.
4. Click the right arrow button to assign.
   The attribute is now listed under the product hierarchy.
5. Change the **Required** and **Default** values as needed. Required attributes must have default values, and will be applied to each assembly in the product hierarchy.

To remove an user-defined attribute

1. From the Product Hierarchy window (see Set Up Assembly Levels (page 116)) menu, select **Tools > User-Defined Attributes...**
2. Select the attribute to be removed from a product hierarchy on the right side of the window.
3. Click the left arrow to remove.

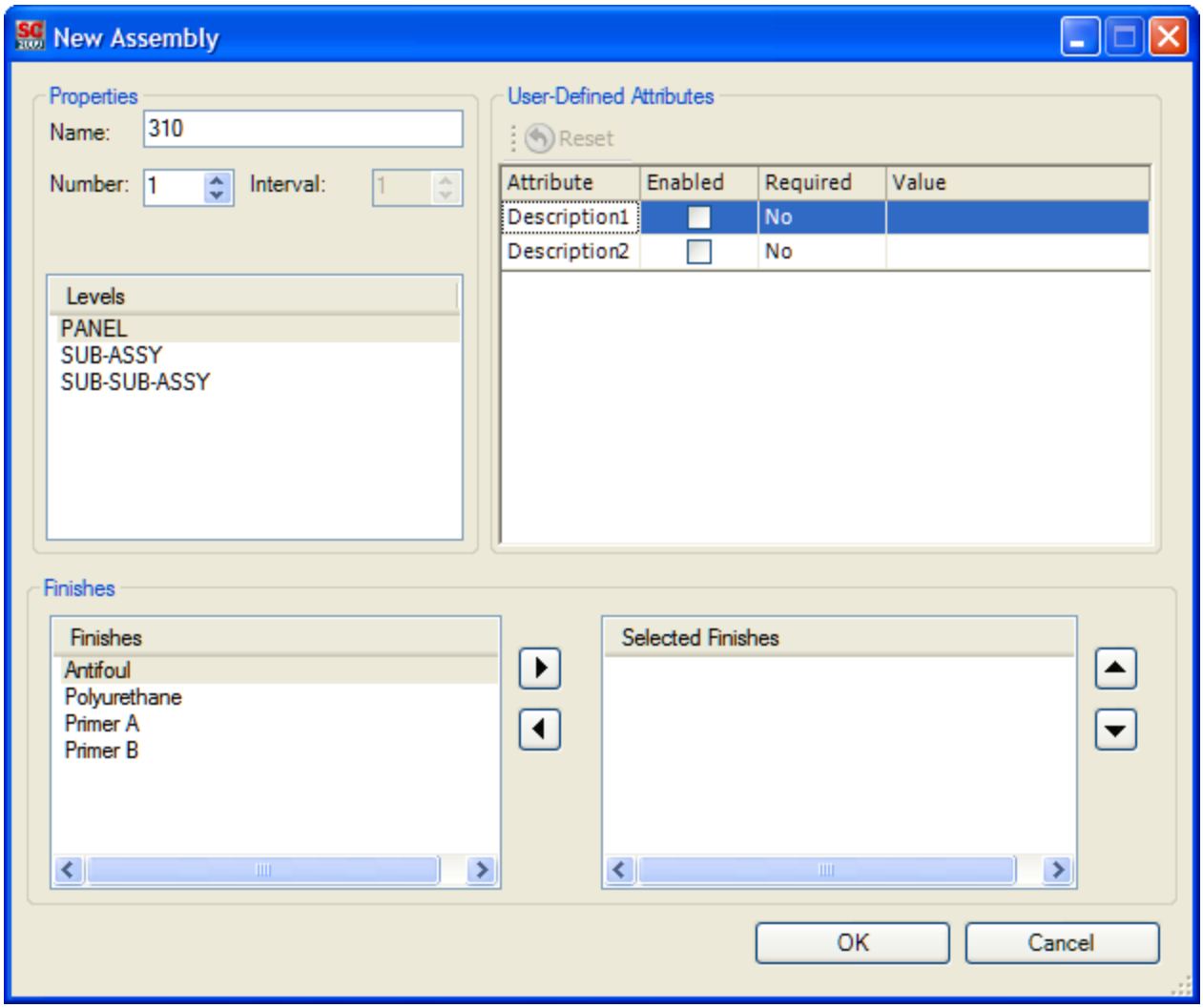
Set Up Assemblies

Add an Assembly to the Product Hierarchy

To add an assembly to the product hierarchy

1. In the Product Hierarchy window (see Set Up Assembly Levels (page 116)), select an assembly to add a sub-assembly to, and choose Tools > Assembly > New. This option is also available if you right-click on the assembly.

   The New Assembly window appears.



2. Set the options:

   - Name – The name of the new assembly.
   - Number – To create more than one assembly, enter the number of assemblies to create. (If you set Name to FR122 and set Number to 3, ShipConstructor will create assemblies FR122, FR123, and FR124.) To create one assembly, set Number to 1.
   - Interval – If Number is greater than 1, Interval is the interval between the numbers used in the assembly names.
   - User-Defined Attributes – You may specify values for any user-defined attributes assigned to the current Product Hierarchy (see Set Up User-Defined Attributes (page 117)). To specify a value, select the Attribute, make sure attribute is enabled with a check in the checkbox, then type a value in the Value field. Attributes can be reset to their default value by clicking on the “Reset” button.

- Level – The level for the new assembly. (The list consists of assembly levels below the assembly you selected in the Product Hierarchy window.)
- Finishes – All finishes.
- Selected Finishes – Finishes assigned to the assembly. Select a finish from the Finishes list. Click the left arrow button to assign the finish to the assembly. You can use the up and down arrow buttons to reorder the finishes. Click the right arrow button to unassign a finish.

3. Click OK.

Rename an Assembly

To rename an assembly

1. In the Product Hierarchy window (see Set Up Assembly Levels (page 116)), select the assembly you want to rename and choose Tools > Assembly > Rename. This option is also available if you right-click on the assembly. Unit-level assemblies cannot be renamed.

   The assembly name in the tree is now editable.

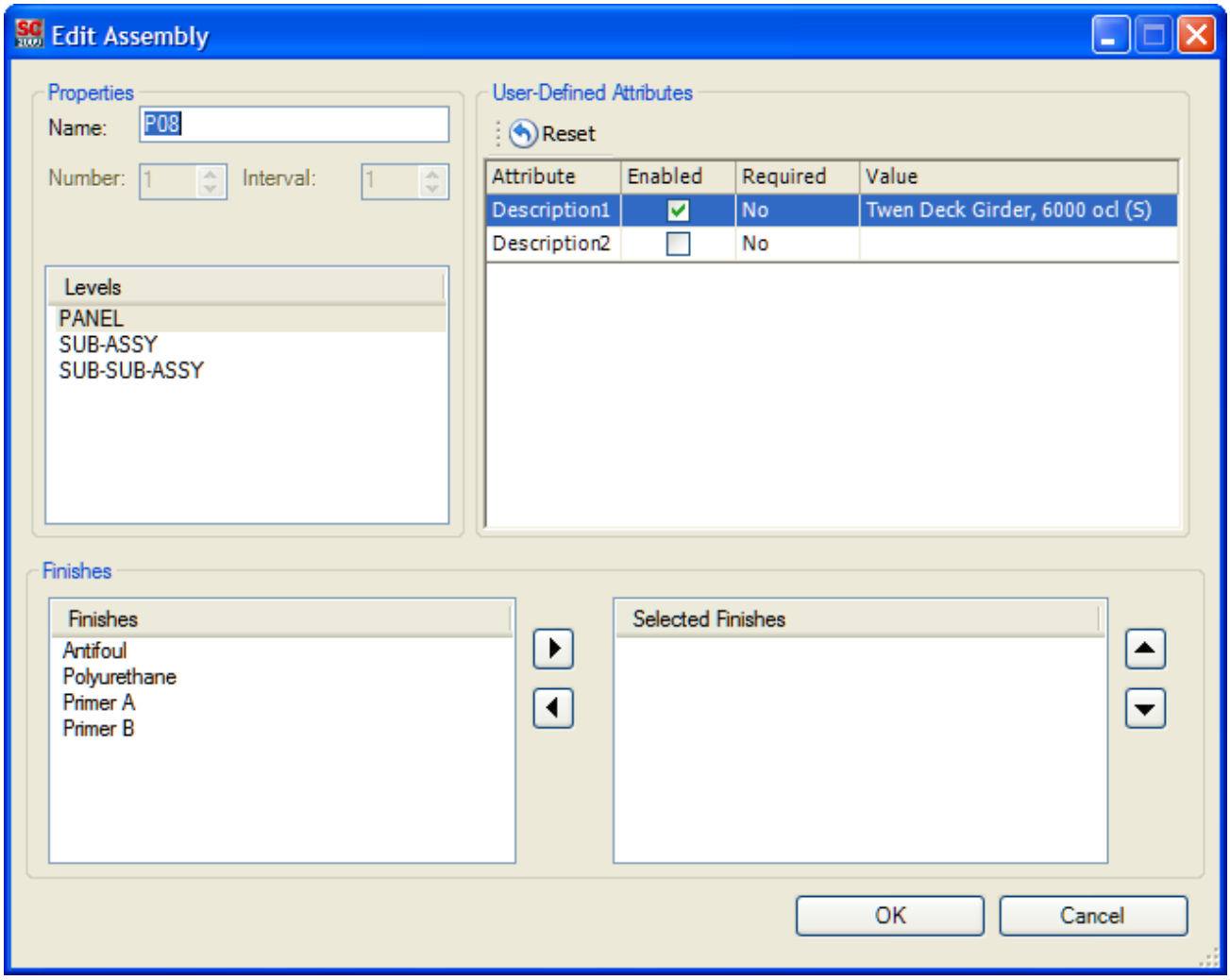
2. Enter a new name and press Enter.

Edit an Assembly

To edit an assembly

1. In the Product Hierarchy window (see Set Up Assembly Levels (page 116)), select the assembly you want to edit and choose Tools > Assembly > Edit. This option is also available if you right-click on the assembly.

   The Edit Assembly window appears.



2. Change the options.
3. Click OK.

Delete an Assembly

To delete an assembly

1. In the Product Hierarchy window (see Set Up Assembly Levels (page 116)), select the assemblies you want to delete.
2. Choose Tools > Assembly > Delete.

   ShipConstructor deletes the assemblies.

   If the assembly contains parts, ShipConstructor re-assigns the parts to the parent of the deleted assembly.

   If the assembly contains sub-assemblies, ShipConstructor deletes the sub-assemblies.

Change the Level of an Assembly

You may want to change the level of an assembly if you are adding a new assembly level but have already created some assemblies and need to move them up or down within the product hierarchy. If an assembly has the level directly below the level of its parent, and its level is the level directly above the level of its children, its level cannot be changed.

To change the level of an assembly

1. In the Product Hierarchy window (see Set Up Assembly Levels (page 116)), select the assemblies and choose Tools > Assembly > Edit or right-click on the assembly.

The Edit Assembly window appears.

2. Select the new level for the assemblies in the Level list.
3. Click OK.

Copy an Assembly

To copy an assembly to another assembly

1. In the Product Hierarchy window (see Set Up Assembly Levels (page 116)), select the assemblies you will copy and choose Tools > Copy Assemblies. This option is also available if you right-click on the assembly.
2. Select the assembly to paste the copied assemblies to and choose Tools > Paste Assemblies.

Assign Parts, Spools, or Assemblies to Assemblies

All three different types of objects (assemblies, spools, and parts) in the Product Hierarchy are assigned to an assembly. These assignments can be modified by assigning those objects to another assembly. There are two methods to do this.

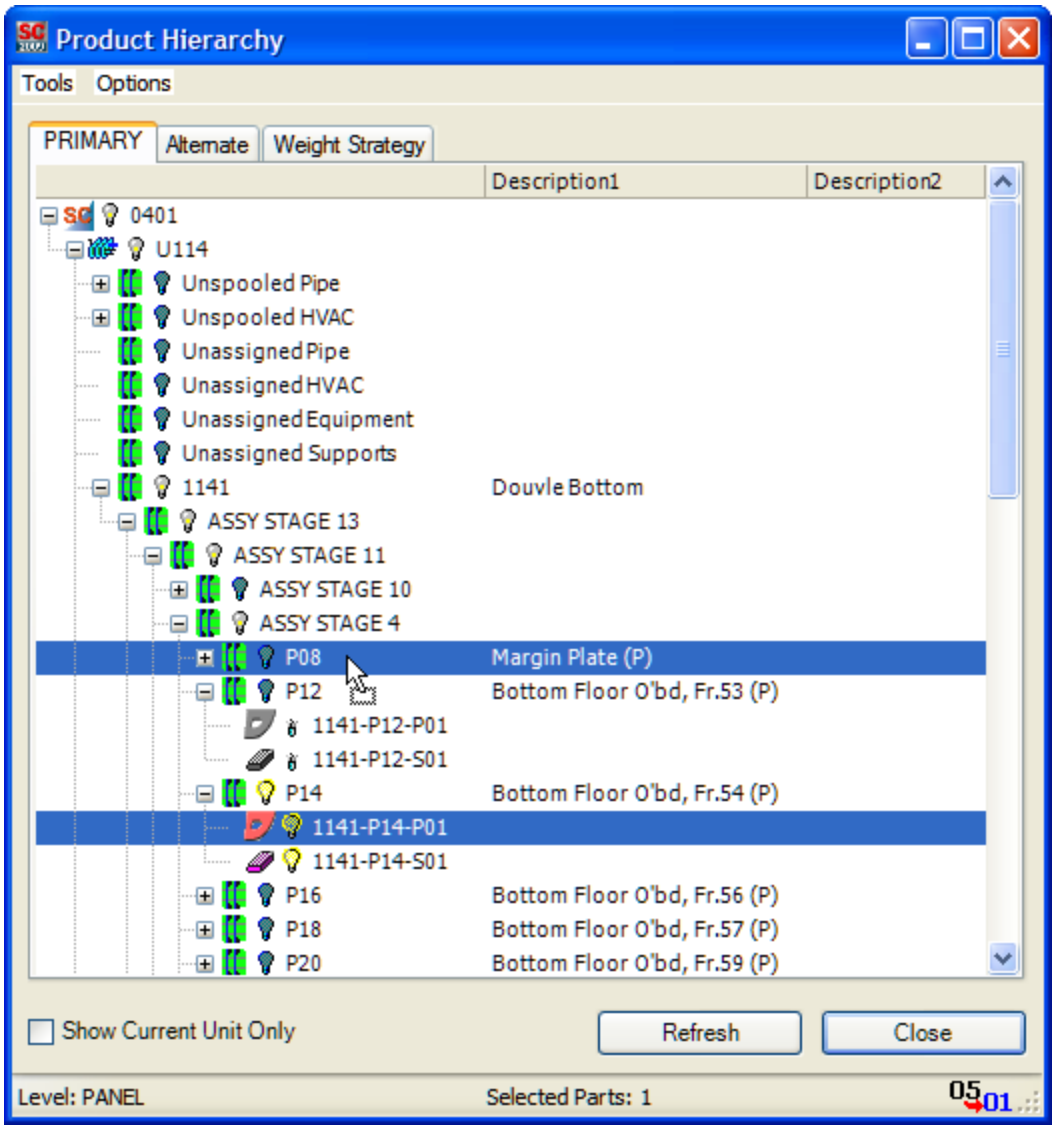
To assign parts or assemblies to an assembly (method 1)

Note: You cannot undo (Edit > Undo) a product hierarchy assignment.

1. Select the parts (or spools or assemblies) in the current drawing or in the Product Hierarchy window.

   If you select parts in the drawing, they are automatically selected in the Product Hierarchy window, and vice versa.

   You can select multiple parts by holding down the Shift or Ctrl key.
2. In the Product Hierarchy window, drag and drop the selected objects onto the target assembly to assign them to.



To assign parts to an assembly that is beyond the limits of the Product Hierarchy window, while dragging the parts, move the cursor above or below the Product Hierarchy window to scroll the display within the Product Hierarchy window up or down. (To scroll to the top or bottom of the Product Hierarchy window, hold down the Ctrl key while dragging. To scroll one page at a time, hold down the Shift key.)

You can assign an assembly to another assembly only if the moved assembly is below the level of the target assembly.

To assign parts to an assembly (method 2)

Note: You cannot undo (Edit > Undo) a product hierarchy assignment.

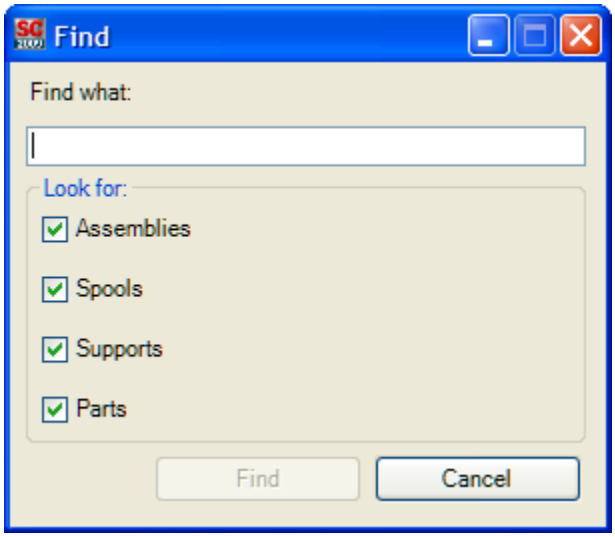
1. Select the parts and/or assemblies to reassign.
2. Choose Tools > Assign to Assembly.
3. The Select Assembly window appears.
4. Select the assembly to assign the object to.
5. Click OK.

Find, Zoom, Remove, Hide, and Show Parts

To find a part

1. In the Product Hierarchy window (see Set Up Assembly Levels (page 116)), choose Tools > Find. This option is also available if you right-click on an assembly. The Find window can also be used to search for assemblies and/or spools.

   The Find window appears.



2. Enter the full or partial name of the part.
3. Click OK.

   ShipConstructor selects the part.
4. If the slected part is not the one you were searching for, you can choose Tools > Find Next (or press F3) to select the next part in the tree that contains the specified search word.

To zoom in to a part in the drawing

1. In the Product Hierarchy window (see Set Up Assembly Levels (page 116)), select the parts or assemblies to zoom in to.
2. Right-click on any of the selected parts or assemblies and choose Zoom To.
3. If the parts are in the current drawing, ShipConstructor will zoom the drawing view so the parts fill the view.

To remove parts from a drawing

> Warning: If you remove parts from a planar group drawing or curved group drawing, the parts will be deleted.

1. In the Product Hierarchy window (see Set Up Assembly Levels (page 116)), select the parts or assemblies.
2. Right-click on the selected parts and assemblies and choose Unload.

   ShipConstructor removes the parts from the current drawing.

To hide or show parts in the drawing

1. In the Product Hierarchy window (see Set Up Assembly Levels (page 116)), click the light bulb beside the part or assembly.
   - Part is in the current drawing and it is visible.
   - Part is in the current drawing but is not visible (Hidden).
   - Part is not in the current drawing.

- Assembly or spool with some hidden parts and some visible parts

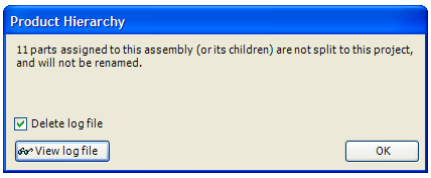
When you click on the light bulb, ShipConstructor changes the part's (or assembly's) visibility in the current drawing. You can also use SC Utilities > Hide Objects, SC Utilities > Unhide Objects, and SC Utilities > Unhide All Objects to hide or show (unhide) parts.

Renaming Parts

Parts with generated names can be renamed through a variety of methods. The most common one is being assigned to a new assembly. Renaming, or changing the level of, an assembly that has parts assigned to it (or any of its children) will also regenerate the names of those parts. Names will only be generated from the part assembly assignments in the primary product hierarchy. Parts with user-defined names (such as Standard Parts) will never be renamed.

Parts will be renamed only when the current user owns the parts for that unit and their naming convention requires it. Only Structure parts and Hangers are affected by this, because the other part types do not allow assembly elements in their naming conventions.

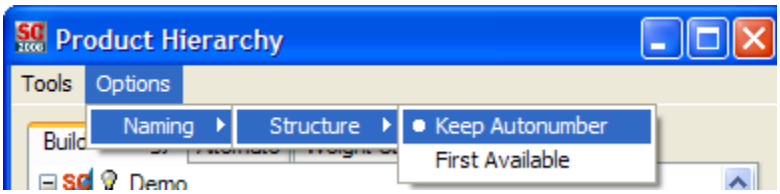
If a project is split in such a way that separate users have control of a unit's parts and that unit's product hierarchy, then, after modifying the product hierarchy, those parts, with naming conventions indicating that the parts need to be renamed due to the modification, will not be renamed. If the user makes a change to the product hierarchy tree such that parts that he does not own will be modified, a window will appear warning him of this, and optionally listing the affected parts.


Product Hierarchy
11 parts assigned to this assembly (or its children) are not split to this project, and will not be renamed.
Delete log file
View log file
OK

By default, when parts are renamed, they use their current autonumber: the elements in that name related to their location in the product hierarchy will be modified. However, some users prefer to model all their parts before defining the complete product hierarchy, and only assigning those parts later. This can cause parts to have exceptionally high autonumbers.

To allow the user to have their parts use the lowest number possible, there is an option for renaming. This option is available only to Structure.

In the Options menu, under Naming > Structure, there are two choices: Keep Autonumber and First Available. The default is Keep Autonumber. In the bottom right corner of the Product Hierarchy window, there is an image indicating the current naming option: Keep Autonumber uses a "05 -> 05" image, and First Available uses a "05 -> 01" image.



Keep Autonumber does just that: parts will try and keep their autonumber when renamed. If that autonumber is unavailable (there may already be a part at the target assembly with that number), then it will look at all the identical parts for the part and try use their autonumbers, and if none of those are available, then it will use the next available number.

First Available will make the parts use the lowest autonumbers possible. If the part has identical parts, with generated names, already at the target assembly, then find the lowest autonumber from that group, and use that number for the part being moved. If the part doesn't have identical parts at the target assembly, or none of the identical parts have generated names, then find the next available autonumber, and use the lower between the next autonumber and the part's current autonumber. If that number is unavailable, then it will use the autonumber of an identical part, if it has any.

Structure Modeling Concepts

ShipConstructor now stores all geometry, parts, and so on, and their attributes (for example, material, weight, revisions, and product hierarchies) within the project database. Whenever you open a drawing, ShipConstructor regenerates the drawing using the information in the database. Previously, ShipConstructor stored geometry within the drawings.

Changes you make to a drawing are not saved to the database until you save the drawing. In ShipConstructor2005 the database is updated as you work in a structural drawing. If two users are editing different versions of a common (identical or mirrored) part, the first user to save their drawing has their edits saved to the database; the second user is notified of a concurrency issue. In ShipConstructor changes that you make to a drawing are not saved to the database until you save the drawing.

Import a Hull Model from ShipCAM

Geometry that is used to start structural modeling can be imported into ShipConstructor from a variety of programs. ShipCAM files can be imported directly into a 3D unit drawing.

To import a hull model from ShipCAM

1. Open the 3D unit drawing. See Create Units (page 107).
2. Choose SC Utilities > ShipCAM > Import Files (page 468)
3. Select Frame, Waterline, Buttock, or Skewed sections files from the ShipCAM project folder.
4. Check that the position and scale of the geometry is correct before proceeding.

Planar Groups

A planar group is a collection of parts that all lie within the same plane (for example, a frame, a bulkhead, a girder, a deck, or a face on a deckhouse). Planar groups are also referred to as planar group drawings, 2D planar groups, 2D planar group drawings, and group drawings.

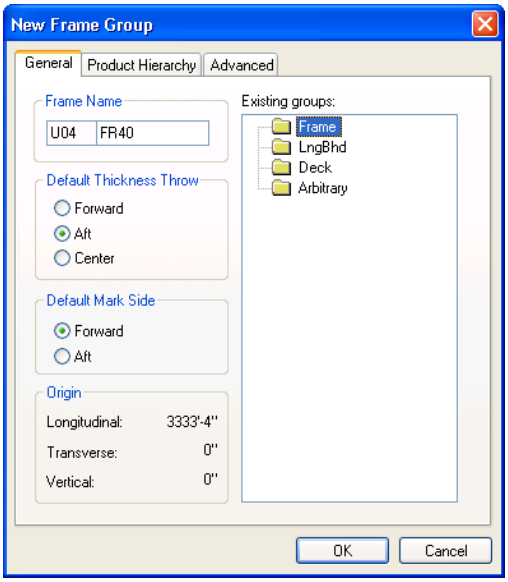
In general, you set up a planar group for each major structural planar object by cutting sections through the hull surfaces. After you create planar groups, you work within each planar group (in 2D) to create the parts for that planar group.

When you create a planar group, ShipConstructor generates a user coordinate system (UCS) on the molded line of the planar group named <drawing_name>_M (for example, U12F112_M). For smaller parts, such as brackets, it is more convenient to create new UCSs and define parts on those UCSs rather than creating planar groups for all the planes of the brackets.

There are three ways to create a planar group:

- Within a unit drawing (as in ShipConstructor2005)
- Within a hull drawing (new in ShipConstructor 2006)
- Within a planar group drawing (new in ShipConstructor 2006)

When you create a planar group, ShipConstructor generates a UCS on the molded line of the planar group and you can create additional custom UCSs to use when you mark group intersections from other planar groups.



When creating a planar group, the ShipConstructor New Frame Group window has been modified as follows:

- The default plate thickness has been removed from the General tab. (When defining individual parts you will specify the stock that then defines the solid.)
- The new ProductHierarchy tab lets you set a default assembly for parts created within the planar group. (When creating a part, you can override the default and assign the part to any assembly.)
- The new Advanced tab lets you set the markline style and create datum lines on planar groups, for example, to help align plates. (Previously, creating datum lines was separate from creating planar groups.)

Create a Planar Group Drawing Template

You can control the appearance of planar group drawings by setting up a planar group drawing template.

To create a planar group drawing template

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.
2. Select the Templates page.
3. Select the Structure folder.
4. Click New Structure.

   The New Drawing window appears.
5. Enter a name for the planar group drawing template.
6. Click OK.

Create Planar Group Drawings

You can create planar group drawings from within a unit drawing or Hull drawing. You can also create a planar group drawing from an existing 2D structural drawing (for example, if you have 2D design drawings containing typical frames, decks, and so on).

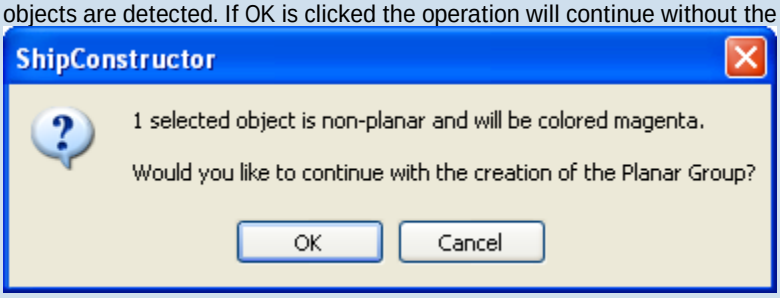
To create planar group drawings

1. From within a unit drawing, hull drawing, or planar group drawing, do either of the following:
   - Choose ShipConstructor > Planar Group > Create Planar Groups.
   - In Navigator, select the Structure page and click New.
2. Select the objects to create planar groups from (for example, sections cut through hull surfaces, such as a frames, decks, or girders).

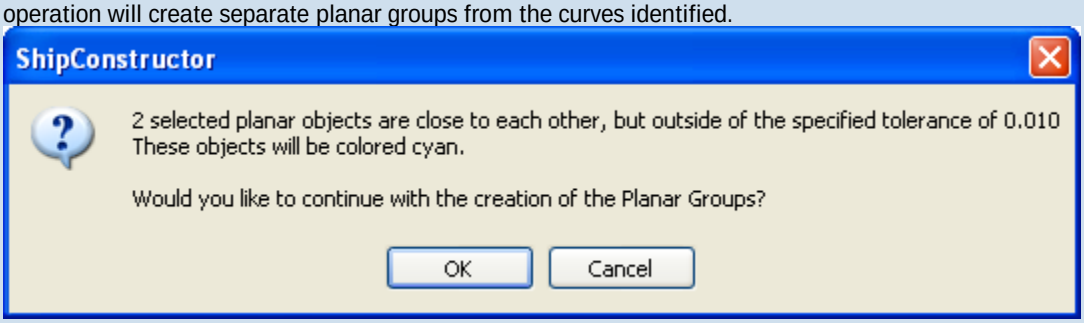
3. Press Enter.

   ShipConstructor checks if each object is planar and determines whether the planar groups are decks, frames, longitudinal bulkheads, or arbitrary groups, based on the orientation of the selected objects. (An arbitrary group is one that is oriented in any direction, such as a canted frame, a girder not parallel to the centerline, or a sloping deck).

   Note: If you are creating a planar group drawing based on geometry imported from another CAD system, the geometry may not be as precise as geometry created within ShipConstructor. ShipConstructor uses a precise internal tolerance value for determining the planarity of an object, and a warning will be displayed if non-planar objects are detected. If OK is clicked the operation will continue without the non-planar curves.

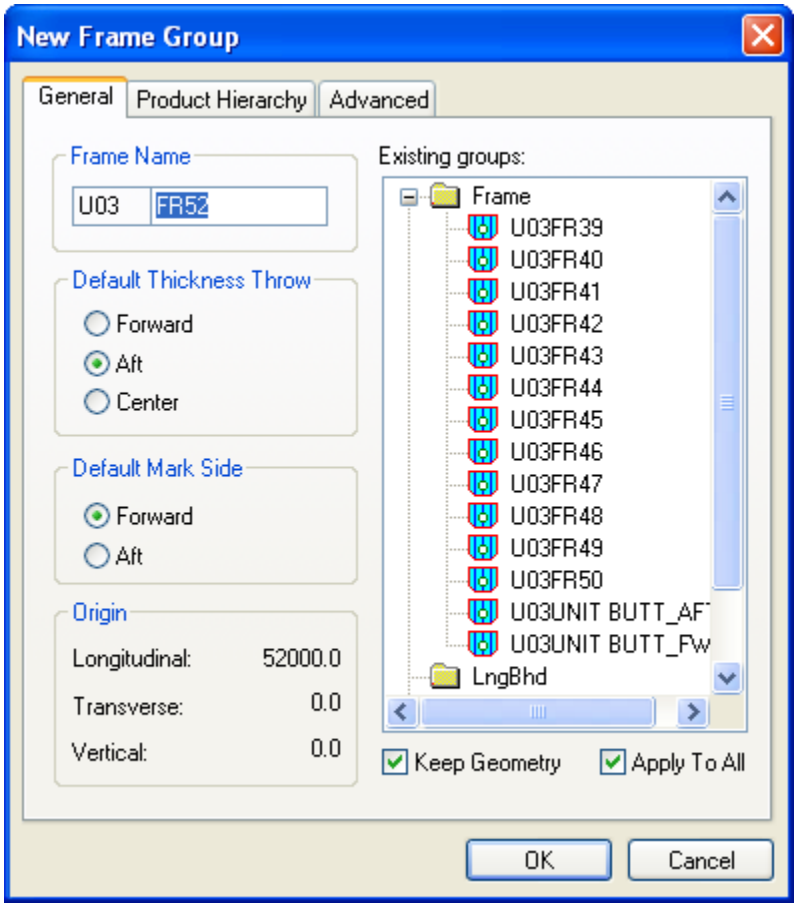
   
   ShipConstructor
1 selected object is non-planar and will be colored magenta.
Would you like to continue with the creation of the Planar Group?
OK
Cancel

   Alternately, if planar curves are detected close to each other but outside of the Planar Group Tolerance value defined in Manager > General > Project Settings > Structure, a warning will be displayed. If OK is clicked then the operation will create separate planar groups from the curves identified.

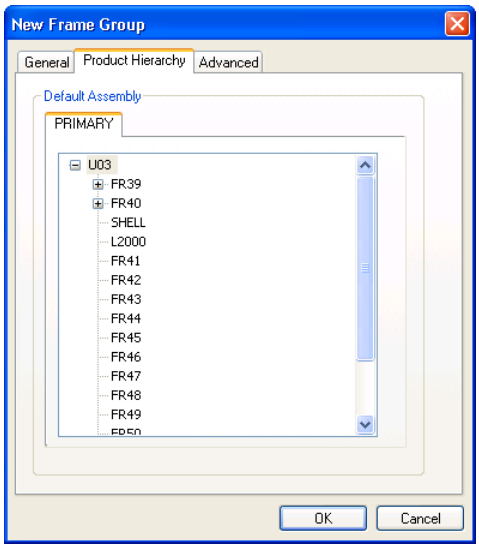
   
   ShipConstructor
2 selected planar objects are close to each other, but outside of the specified tolerance of 0.010
These objects will be colored cyan.
Would you like to continue with the creation of the Planar Groups?
OK
Cancel

   In either of the above cases it is recommended to use the SC Utilities > 3D to 2D command on the curves for each planar group before creating the new planar group (in order to ensure all curves are planar and on the same plane).

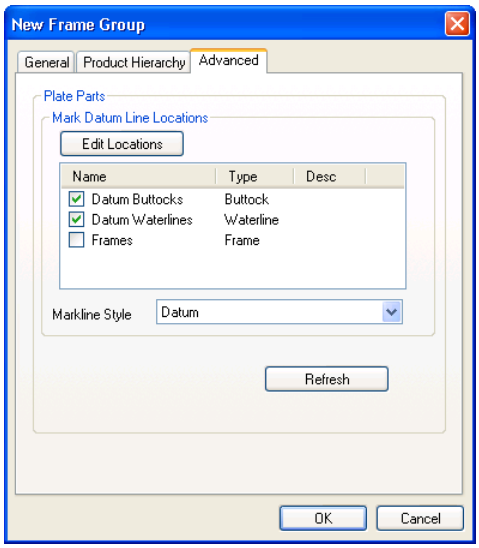
   The New Frame Group window appears.



4. Set the options:
   - Name – ShipConstructor suggests a name for the planar group drawing based on the unit name and the layer name of the first selected object. ShipCAM and some other hull fairing programs can place exported objects onto named layers. Ideally, the layer names should indicate the name of a group, for example FR_110.
   - Default Thickness Throw – When creating plates in this planar group this is the default thickness direction. Only the first part created in this planar group will default to this direction. Each subsequently created part will use the throw direction of the last created part by default.
   - Default Mark Side – The side of the plate to mark by default for any plates within this planar group.
   - Keep Geometry – If this is not checked then the geometry used to create the planar groups will be removed after the operation.
   - Apply to All – The selected options (including those in the Product Hierarchy and Advanced tabs) will be applied to all of the planar groups created (with the exception of the name, which is taken from the curves used to create the planar group or the layer the curves reside on if no name is assigned to the curves).

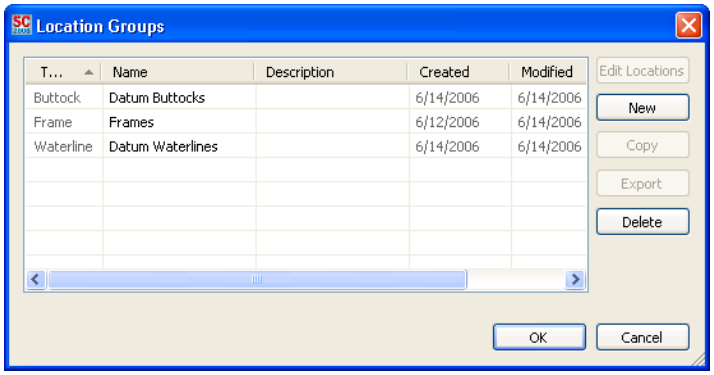


Default Assembly – Lets you set the assembly that is used when you define parts in the planar group. This lets you organize parts before they are made. If modelers are not responsible for assigning parts to the product hierarchy, parts will not end up in one big pool of parts when the build strategist organizes parts. If you do not select a default assembly, the Unit assembly is selected.



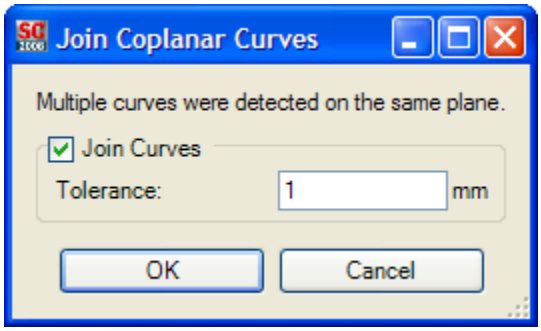
Mark Datum Line Locations – If you use datum lines on parts, enable this option to set up marking on plate parts as they are created.

Edit Locations – Lets you create or edit locations and location groups.



Note: You can later edit these values. See Edit Planar Group Properties (page 133).

5. If the planar group is being created using multiple curves the Join Coplanar Curves window will appear. If you wish to join the curves, check the Join Curves box and specify a tolerance (maximum distance) for them to be joined.



6. Click OK to create the planar group drawing. A log file detailing the results of the operation will be displayed. Ensure you check the log for any errors or warnings from the creation process.

To create a planar group drawing from a 2D structural drawing

For this example, we will suppose that you want to use a 2D structural frame (scantling) drawing.

1. Make a copy of the 2D structural frame drawing.
2. In the copy, erase all objects you do not want. There might be several frames in the same drawing.
3. Make sure that the origin of the frame is at (0,0). This should be in the world coordinate system. Move the frame objects if necessary.
4. We assume that you have a project with a unit already. If not, make a new project and create a unit with an empty drawing.
5. Open the unit drawing appropriate for the project. This drawing might be of a completely new project with nothing in it or it might already have several structural drawings.
6. In the unit drawing, make sure the WORLD UCS is active. Do this by using either UCS or UC on the command line.

   Draw a line at the location of the frame. For example, if the frame is at 50.2 m, draw a line giving the values manually as 50200,0,0 to 50200,5000,0. Remember ShipConstructor drawings are in millimeters or inches.
7. Use the Planar Group - “New” function.
8. Select the line at 50200.

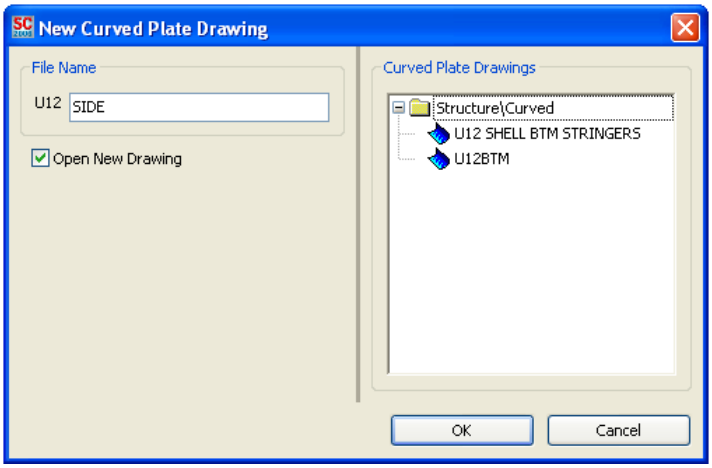
9. It is not obvious to ShipConstructor that the line is a frame. The line could be part of a deck or a frame. In the window select Frame and click OK.
10. You will see the New Frame Group window. Enter a name for the frame and set the throw and thickness. Click OK to create the ShipConstructor structural drawing.
11. Open the ShipConstructor structural drawing. It contains just a single line.
12. Type DDINSERT at the command line.
13. Insert the frame drawing you prepared at the origin (0,0,0) by Specify Parameters on Screen and check the Explode option.
14. The frame drawing is now inserted in 3D space at the correct location and is ready for detailing. You may have some clean up to do.
15. Move objects to the correct draft layers.
16. The drawing you inserted may have created several new layers in the ShipConstructor structural drawing. We do not need them any more. PURGE all unused layers. Do not worry about deleting ShipConstructor layers; they will be recreated when needed.

Create Curved Model Drawings

A curved model drawing is similar to a planar model drawing, except a curved drawing contains curved plates instead of planar parts. You use curved model drawings to add shell plating (in the form of curved and expanded plates). In curved model drawings, all production data of the expanded plate is usually located in the XY plane of the world coordinate system.

To create a curved group drawing

1. In Navigator, select the Structure page.
2. Select the Curved folder in the drawing list.
3. Click New. The New Curved Plate Drawing window appears.



4. Enter a name for the new curved group drawing. Existing curved group drawings are listed on the right side of the window.
5. Click OK.

Edit Planar Group Properties

To edit the properties of a planar group

> Note: Make sure the planar group drawing you want to edit is not currently open by someone else.

1. In a unit drawing, choose ShipConstructor > Navigator (page 368) to open Navigator.
2. Select the Structure page.
3. Select the planar group drawing that you want to edit.
4. Click Properties to open the Edit Planar Group Properties window.
5. Set the options. See Create Planar Group Drawings (page 127).
6. Click OK.

Delete a Planar Group

To delete a planar group

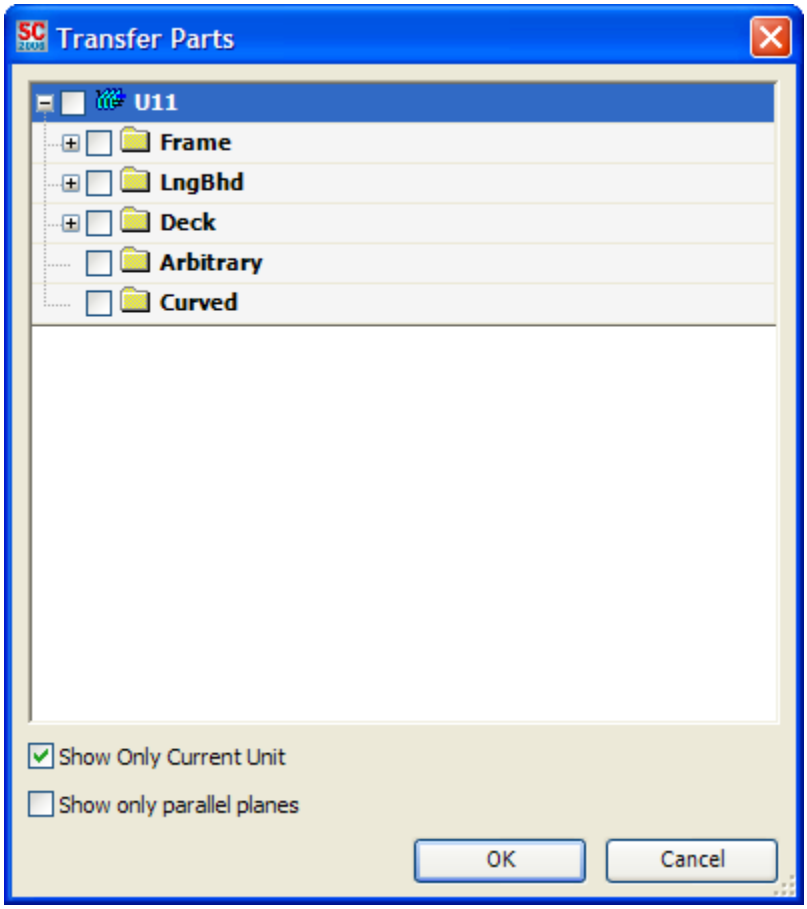
1. Choose ShipConstructor > Navigator (page 368) to open Navigator.
2. Select the Structure page.
3. Select the planar group drawing you want to delete.
4. Click Delete.

Transfer Objects from One Planar Group Drawing to Another

You can transfer objects from the current planar group drawing to any other planar group drawing. Transferred objects maintain their position within the ship's coordinate system. This can be useful if you want to split a planar group drawing into two.

To transfer objects from one planar group drawing to another

1. Select the objects you want to move or copy.
2. Choose SC Structure > Transfer Objects to Group...
3. In the warning window that appears, click Yes to save the current drawing and continue.
4. The Transfer Parts window appears.

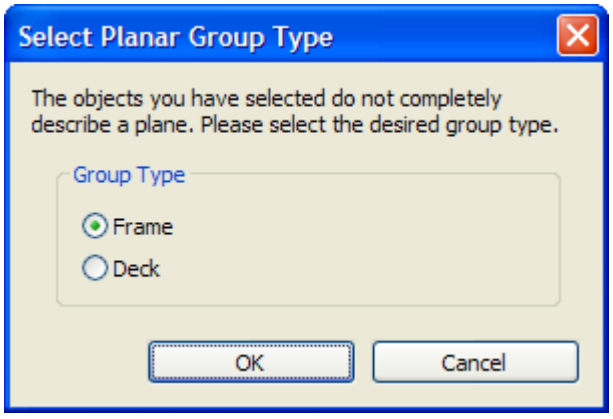


5. Check the box next to the planar group you wish to transfer the selected objects to.
6. Click OK.

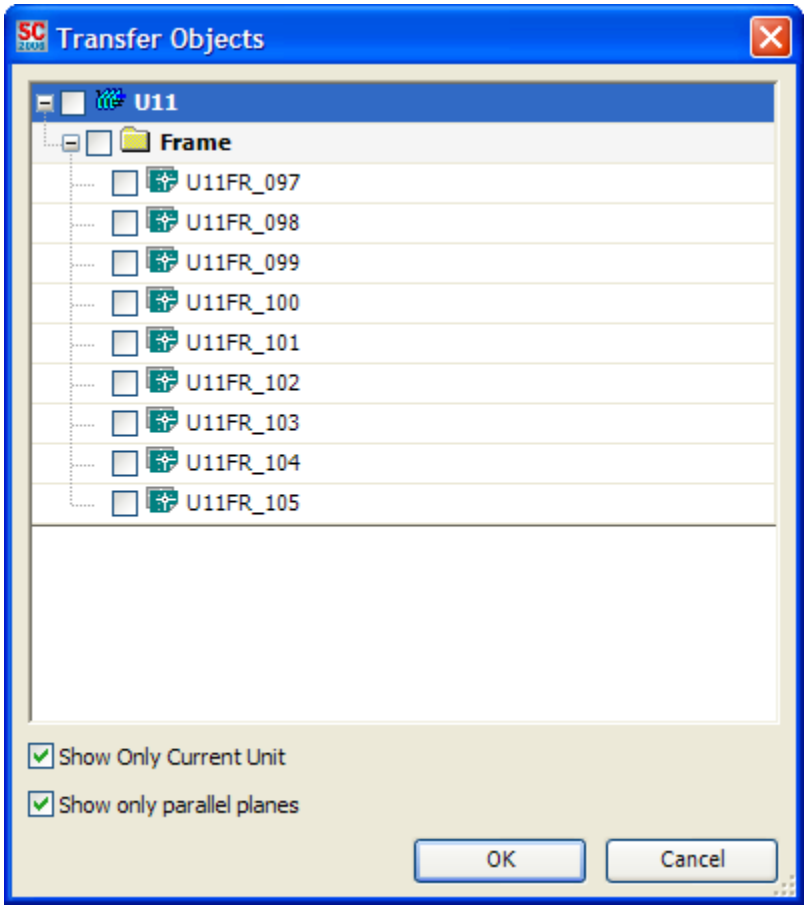
Transfer Objects from a Unit Drawing to a Planar Group Drawing

To transfer objects from a unit or hull drawing to a planar group drawing

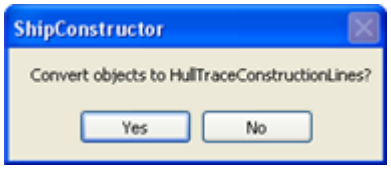
1. Choose ShipConstructor > Planar Group > Transfer Objects to Group.
2. Select the objects to transfer. If the objects do not uniquely identify a planar group type, you will see a window that asks you to select the type.



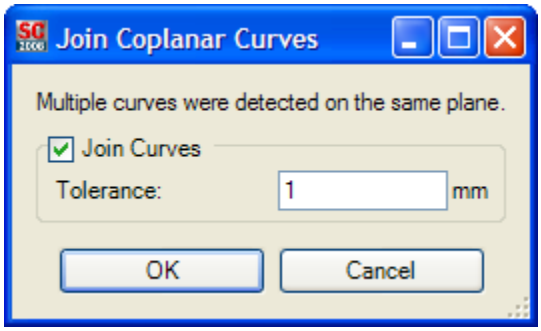
3. In the Transfer Objects window, check the box next to the planar group that you wish to transfer the objects to and click OK.



4. Choose whether you wish to convert the lines to HullTrace construction lines.



5. If the planar group is being created using multiple curves the Join Coplanar Curves window will appear. If you wish to join the curves, check the Join Curves box and specify a tolerance (maximum distance) for them to be joined. Click OK to continue to transfer the objects to the selected planar group.



User Coordinate System (UCS)

AutoCAD UCSs are used throughout ShipConstructor. In the structure model drawings they define the planes of parts. These special UCSs are called planar group planes in ShipConstructor. ShipConstructor will create planar group planes automatically when a planar group model drawing is created. For each planar group plane there is a named UCS. You can create your own named UCSs but they will not be stored in the database unless converted to planar group planes.

Within each drawing there are two coordinate systems.

The world coordinate system (WCS) is a fixed system in which the X axis is horizontal, the Y axis is vertical, and the Z axis is perpendicular to the XY plane. The origin (where the X axis and Y axis intersect) is in the lower left corner of the drawing.

The user coordinate system (UCS) is a system that you can move and rotate (relative to the WCS) to make it easier to work on specific parts of your drawing. You can set up any number of UCSs within a drawing, but only one UCS can be active (this is called the current UCS).

For more information on user coordinate systems, see UCS in AutoCAD Help.

ShipConstructor has these UCS options:

- Dwgname_M (for moldline) – System-generated UCS at the planar group molded line (the same as Dwgname_C in ShipConstructor2005).
- Dwgname_P+Dist (for parallel) – User-defined UCS parallel to the planar group molded line. This is similar to Dwgname_T in ShipConstructor2005, except you can have more than one system per planar group. For example, for the actual (not default) thickness side of parts there are two limits: Dwgname_P+20 would be 20 mm forward, or up, from the molded line and Dwgname_P-20 would be 20 mm aft, or down.
- Dwgname_U – User-defined UCS:
  - Dwgname_U_WL4000 – User-defined orthogonal UCS. For example, if you are within a frame drawing and create a deck UCS, ShipConstructor will name it Dwgname_UWL4000 (for waterline). However, if you are within a deck drawing and create a deck UCS, ShipConstructor will name it Dwgname_P+Dist (for parallel).
  - Dwgname_U_001 – User-defined arbitrary UCS such as skewed planes. By using the ShipConstructor Activate UCS command, the resulting plane is recognized by ShipConstructor and can later be referenced.

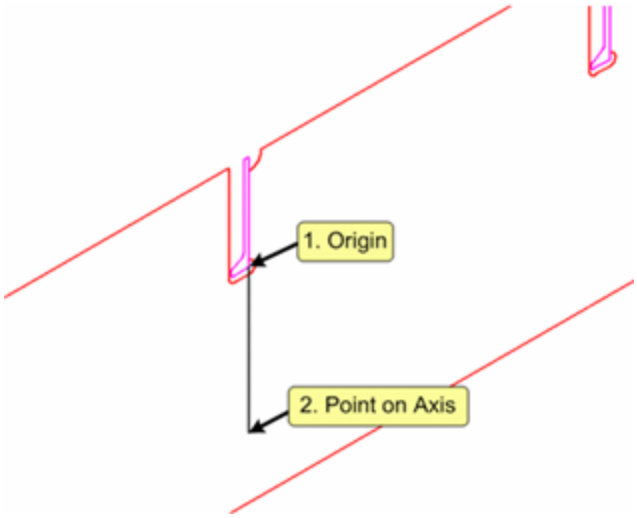
ShipConstructor stores UCSs in the database as planar group planes. Any UCSs that are created manually are not stored in the database.

Create a UCS Out of Plane with the Current UCS

To create a UCS out of plane with the current UCS

1. Choose SC Utilities > Activate UCS (page 451) to open the UCS window.
2. Click New Out of Plane.
3. Click the origin point.

   OSNAP END is automatically set.
4. Select a point along one axis of the UCS.

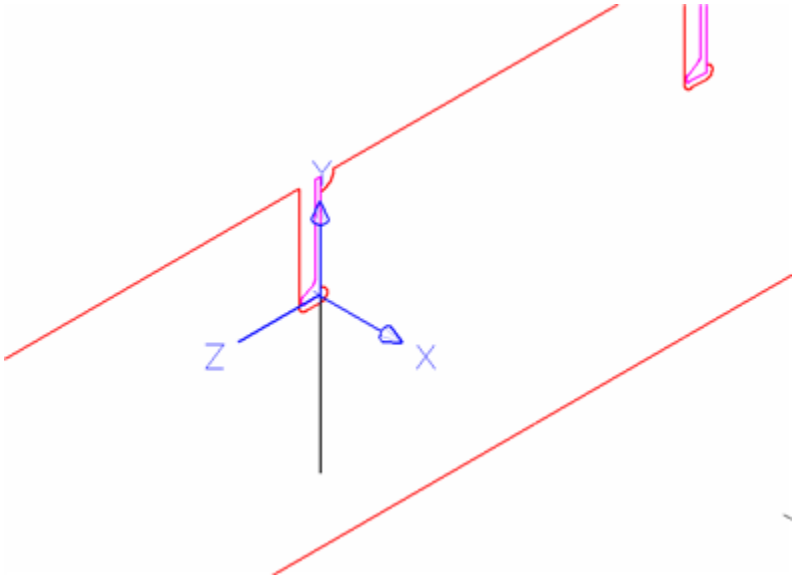


OSNAP END and NEAR are automatically set.

5. To create a UCS perpendicular to the current one, press Enter.

   To create a skewed UCS, click a third point.

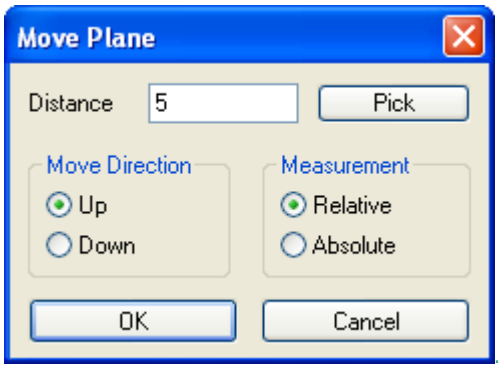
   ShipConstructor creates and names the UCS using the available UCSs.



Create a UCS Parallel to the Current UCS

To create a UCS parallel to the current UCS

1. Choose SC Utilities > Activate UCS (page 451) to open the Activate UCS window.
2. Select the UCS in the list that you want the new UCS to be parallel to.
3. Click Move Z



The Move Plane window appears.

4. Enter a distance, direction and measurement method to offset the new UCS from the selected UCS or click the Pick button to select the position of the new UCS

   ShipConstructor creates a new UCS using the values set by the user or the picked location.

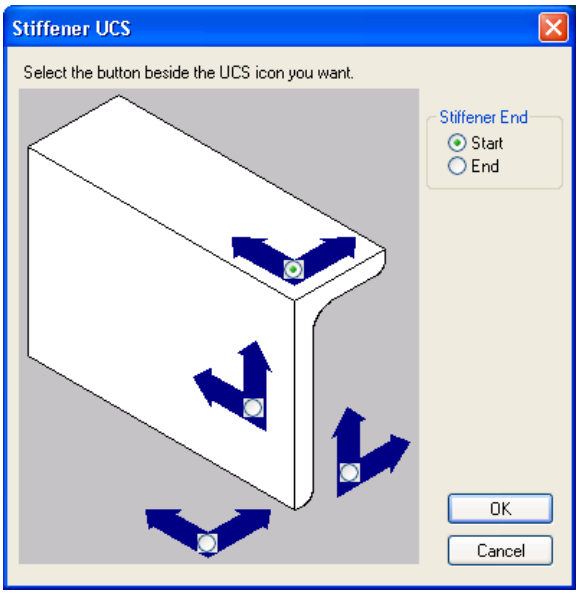
Activate a UCS

To activate a UCS from an object

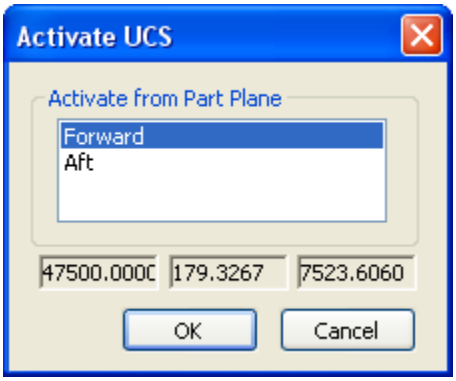
1. Choose SC Utilities > Activate UCS (page 451) to open the UCS window.
2. Click Activate from Object.
3. Select the object.

   ShipConstructor calculates the UCS differently for different types of objects:

   - Standard AutoCAD objects – ShipConstructor determines the UCS plane of the object.
   - ShipConstructor parts – ShipConstructor lets you choose the UCS:
     - Stiffeners – ShipConstructor displays the following window, letting you select one of four UCSs associated with an end of a stiffener.



     - Plates – ShipConstructor displays the following window.



To activate a UCS

1. Choose SC Utilities > Activate UCS (page 451) to open the UCS window.

   Select the UCS from Activate from list.

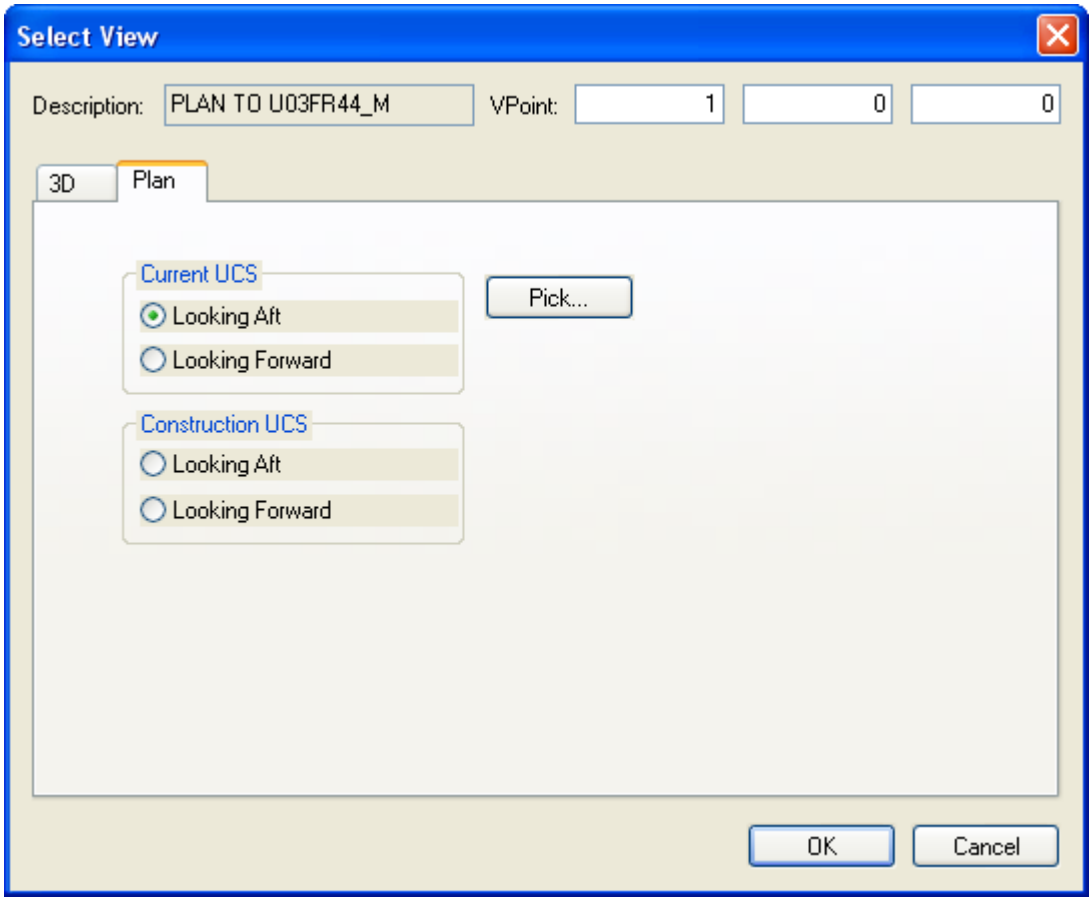
2. Click Activate.

Select a UCS Viewpoint

Selecting a UCS viewpoint is the easiest way to obtain a plan view to the currently active UCS, especially if the UCS is canted, such as for a sloping deck.

To select a UCS viewpoint

1. Choose SC Utilities > 3D Viewpoint (page 451) to open the Select View window.
2. Click the Plan tab.



Views for the current UCS are listed under Current UCS.

If you are currently in a planar group drawing, the views for the construction UCS are listed under Construction UCS.

Note: The view names are based on the orientation of the UCS, for example, Looking Aft.

3. Select a view.
4. Click OK.

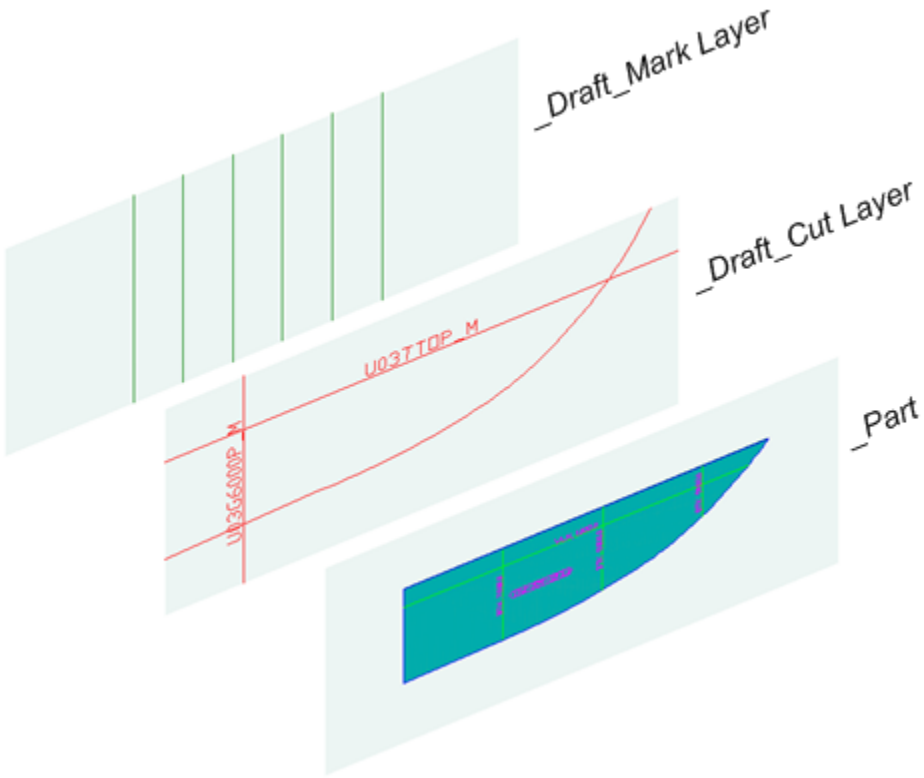
Undo

With AutoCAD you can backtrack your recent actions using one of several methods. For details, see Undo in AutoCAD Help.

Structural Modeling Layers

Structural modeling layers in ShipConstructor are as follows:

- _Part contains the part (now a ShipConstructor custom entity and no longer an AutoCAD block) and its automatically generated solid.
- _REV contains revision information (as in ShipConstructor2005).
- _Draft_Cut contains the outer and inner contours of the part.
- _Draft_Mark contains geometry to be marked on plate parts.
- _Draft_NoProcess contains geometry that will be ignored during NC cutting (as in ShipConstructor2005).



Activate a Layer

When you activate a layer, the objects on the layer become visible and available to edit.

To activate a layer

1. Choose SC Utilities > Layer > Activate (page 459) to open the Select Layer to Activate window.
2. Select the layer you want to activate.
3. Click OK.

Deactivate a Layer

When you deactivate a layer, the objects on the layer become hidden and are unavailable for editing (that is, they become frozen).

To deactivate a layer

1. Choose SC Utilities > Layer > Deactivate (page 459) to open the Select Layer to Deactivate window.
2. Select the layer to deactivate.
3. Click OK.

Copy Objects from One Layer to Another

To copy objects from one layer to another

1. Choose SC Utilities > Layer > Copy Geometry to (page 460)
2. Select the objects you want to copy.
3. Press Enter

   The Copy Geometry to Layer window appears.
4. Select the layer to copy the objects to.
5. Click OK.

Move Objects from One Layer to Another

To move objects from one layer to another

1. Choose SC Utilities > Layer > Move Geometry to (page 460)
2. Select the objects you want to move.
3. Press Enter

   The Move Geometry to Layer window appears.
4. Select the layer to move the objects to.
5. Click OK.

Object Display

Hide or Show Objects

Sometimes it is difficult to select an object when many other objects are near it. You can hide objects to make it easier to see or select the objects you are interested in. This is especially helpful when many objects are on the same layer.

> Note: When you hide an object, ShipConstructor does not move it to a hidden layer. Instead, ShipConstructor simply does not display the object.

To hide objects

1. Select the objects to hide.
2. Choose SC Utilities > Hide Objects (page 454)

To show objects

1. Choose SC Utilities > Unhide Objects (page 454)

   ShipConstructor shows the objects, one at a time, that you most recently hid.

   If you choose SC Utilities > Unhide Objects (page 454) again, ShipConstructor will continue showing objects in the reverse order that you hid them.

To show all objects

1. Choose SC Utilities > Unhide All Objects (page 454)

Display Objects in Wireframe, Hidden Line, or Shaded Mode

You can control whether objects in your drawing are displayed in wireframe, hidden line, or shaded mode.

To display objects in wireframe, hidden line, or shaded mode

1. Choose the mode you want from View > Visual Styles.

Display Information for an Object within an MLinked Drawing

You can list information on an object within a model linked (mlinked) drawing, providing more information for the designer during modeling, checking, or reviewing of a particular area in the 3D product model.

To display information for an object within an mlinked drawing

1. Choose SC Utilities > List Item within Block/Xref (page 466)
2. Select the object within your drawing that you want to see information about.

   The information appears within the command line.

Part Views

A Part View is a snapshot of a part that can be loaded into any drawing to be used for reference or for enhanced display capabilities. Part Views cannot be edited and changes made directly to the Part View will not affect the parts they were created from.

Part View Advantages

- Load a view of any part into any ShipConstructor drawing
- Improved Live Sectioning support
- Flexible to support many loading strategies (ie.Load via Assembly, Extents, Relationships...ect)

Supported ShipConstructor Entities

- All Structure Parts
- All Pipe Parts
- All HVAC Parts
- Equipment
- Supports
- Standard Assemblies (as individual parts)
- Hangers

Licensing

User will need to have a valid Universal, Structure, Pipe, Equipment, HVAC, or Electrical license and up-do-date ShipConstructor subscription to run any of the PartView commands.

Loading Part Views

When Part Views are loaded any parts which currently exist in the current or any M-Linked drawings will not be created as Part Views. If a Part View is loaded into a drawing in which there is already a Part View in the drawing for the same contained part then the existing Part View will be updated and a new Part View is not created.

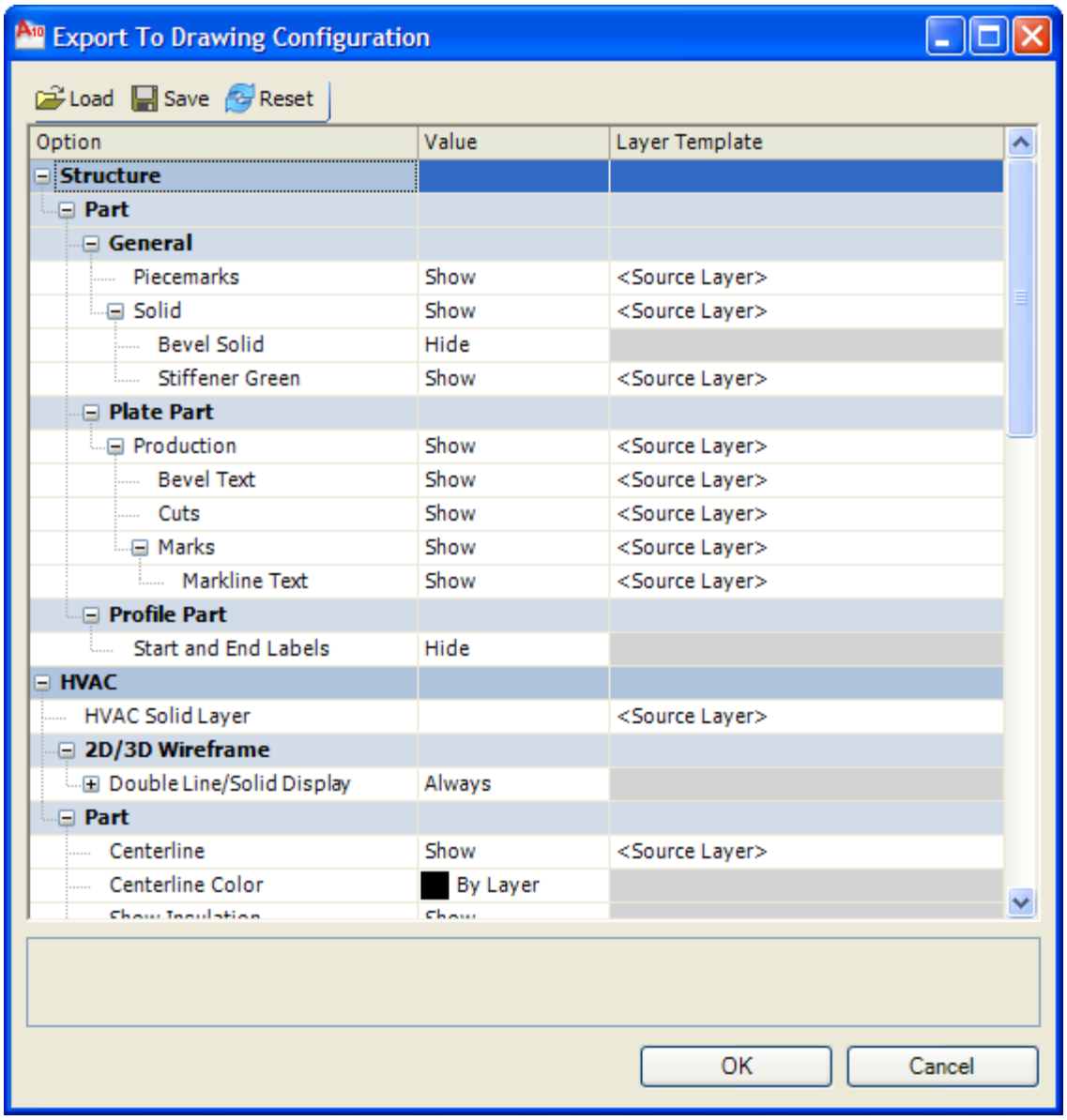
Export

Export a ShipConstructor Drawing to an AutoCAD only Drawing

You may wish to open a ShipConstructor drawing on a machine that does not have ShipConstructor. To facilitate this ShipConstructor has a command that creates a drawing containing only AutoCAD entities from any ShipConstructor drawing.

To export a ShipConstructor drawing to an AutoCAD only drawing

1. Choose the SC Utilities > Export > Export to Dwg... (page 466) menu command.
2. If any changes have been made to the current drawing, you will be asked if you wish to save. Click Yes to continue with the export.
3. The Export To Drawing Configuration dialog will appear. Select the part attributes in the drawing that are to be exported by selecting either Show or Hide under the Value column. Define a layer name for the attribute by overriding the text in the Layer Tempate column. By default, the layer name that is curently assigned to the attribute is exported as indicated by the <Source Layer> text.



4. In the Save As window that appears, choose a location and name for the exported drawing and click Save.
5. The export will proceed and will open up the new AutoCAD drawing when it is complete.

Note: Everything in the ShipConstructor drawing will be exported to the new drawing, including any hidden objects or layers.

Note: The export does not take the Structural Display Options into account; the solids, piecemarks and production info of the parts will all be visible in the exported drawing regardless of the Structural Display Options (similar to how the AutoCAD **EXPLODE** command works for parts). See SC Structure > Show / Hide Options (page 394) for more information on the Structural Display Options.

Export ShipConstructor Drawings to NavisWorks Drawings

It is possible to export the current drawing to a NavisWorks NWC file.

To export ShipConstructor drawings to a NavisWorks drawing

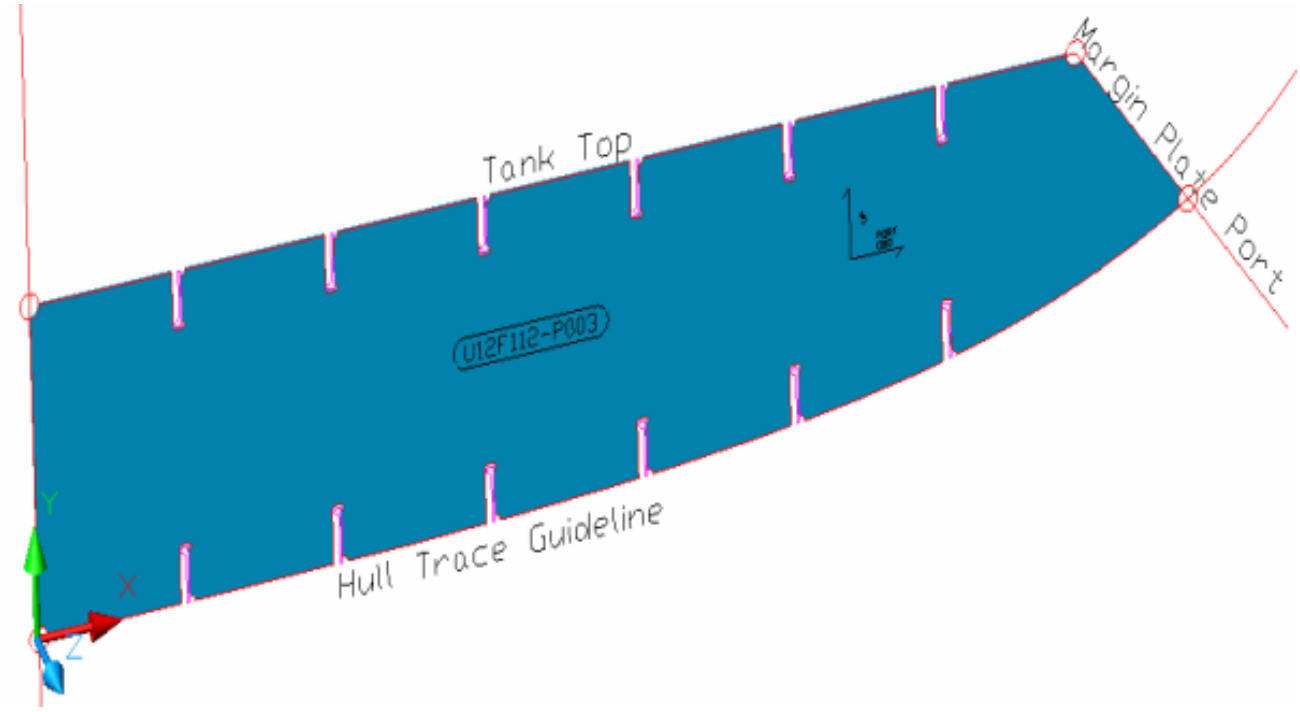
1. Choose SC Utilities > Export > Export to NWC (page 467).
2. If any changes have been made to the current drawing, you will be asked if you wish to save. Click Yes to continue with the export.
3. A window will appear where you can select the drawings from the ShipConstructor hierarchy that you wish to export.
4. Click OK to continue.
5. Each selected drawing from Step 3 will automatically be opened in ShipConstructor and exported to the chosen NavisWorks drawing format.
6. The exported files are created in the same folder as the original drawing with the same name and NWC extension.

Model Structure

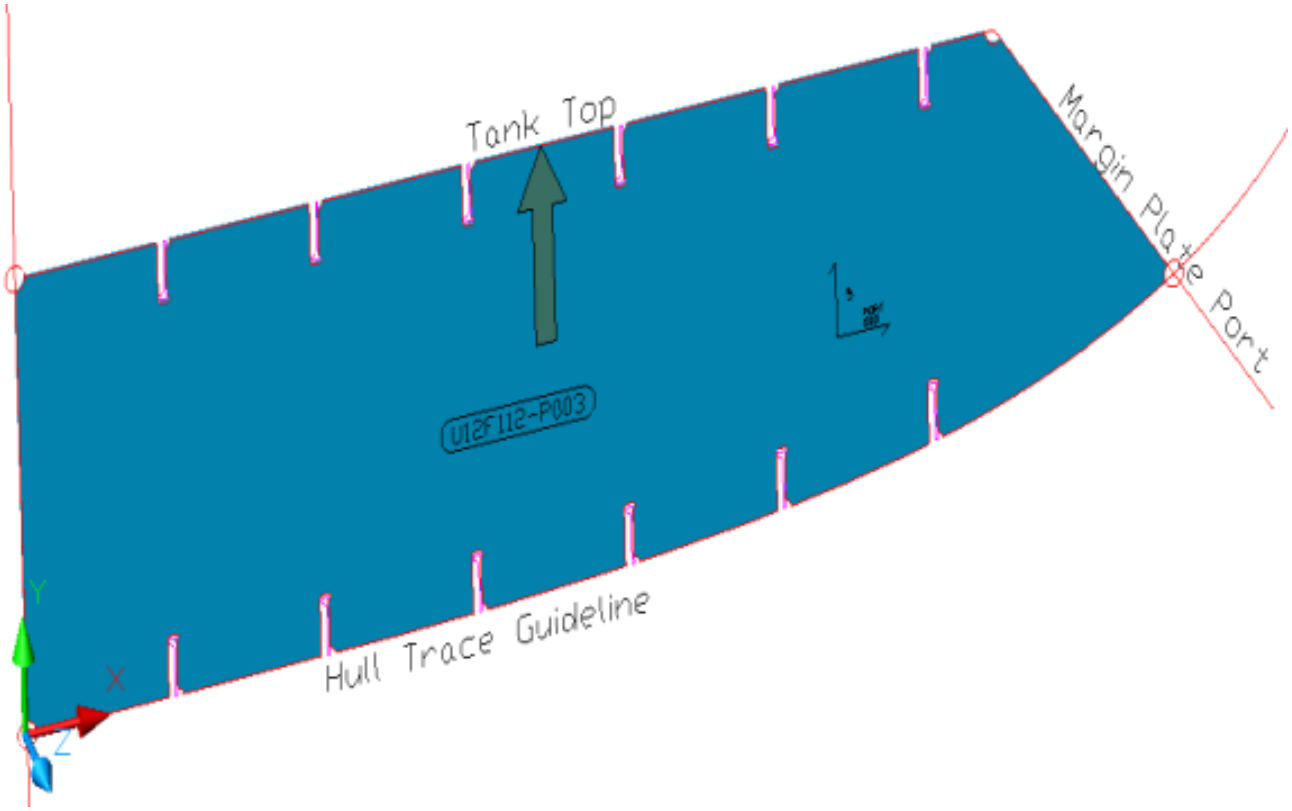
Structural modeling in ShipConstructor now uses a parametric-like workflow. Previously, you had to create solids before creating parts. Now, you can create parts directly from construction lines.

The Database Driven Relational Object Model (DDROM) dramatically reduces the amount of time you spend modeling structural plate parts and significantly simplifies the process for making late design changes. For example, if you move a tanktop up in one drawing, ShipConstructor will adjust the height of all frame plate parts above and below it. Similarly, if you exchange one frame hull trace for another, ShipConstructor automatically updates all related frame parts (plates and stiffeners).

Note: Stiffeners are logically attached to the plate parts and will, therefore, change their location with changes in thickness, throw, or location of the plate part. Subsequent releases of ShipConstructor will also automatically update stiffener lengths.



Moving the tanktop up...



...automatically modifies all related parts

DDROM Best Practices

Conceptual Design Stage

With the ability to quickly create and easily update existing parts in ShipConstructor comes the ability to create macro parts early in the design stage that can then be easily modified or redefined when the design progresses. For example, you can now create a single plate for an entire deck early on in the design stage to allow other disciplines to start using the conceptual model as an early reference model. This deck can then be straked when the structural design requires it to be. Alternatively, due to the speed with which parts can be created and modified, it may make sense to strake the deck early in the process and make any required changes as and when necessary.

Define Parts Early

In sharp contrast to ShipConstructor2005, parts in ShipConstructor can be created as early as necessary, letting you start fleshing out a product hierarchy, perform early weight calculations, and so on when they are required. The existing parts can be detailed later in the process, further refining the weight estimates and detail of the model as you go. In fact, with ShipConstructor you do not start creating a solid model until you begin creating parts, so defining parts is a necessary step.

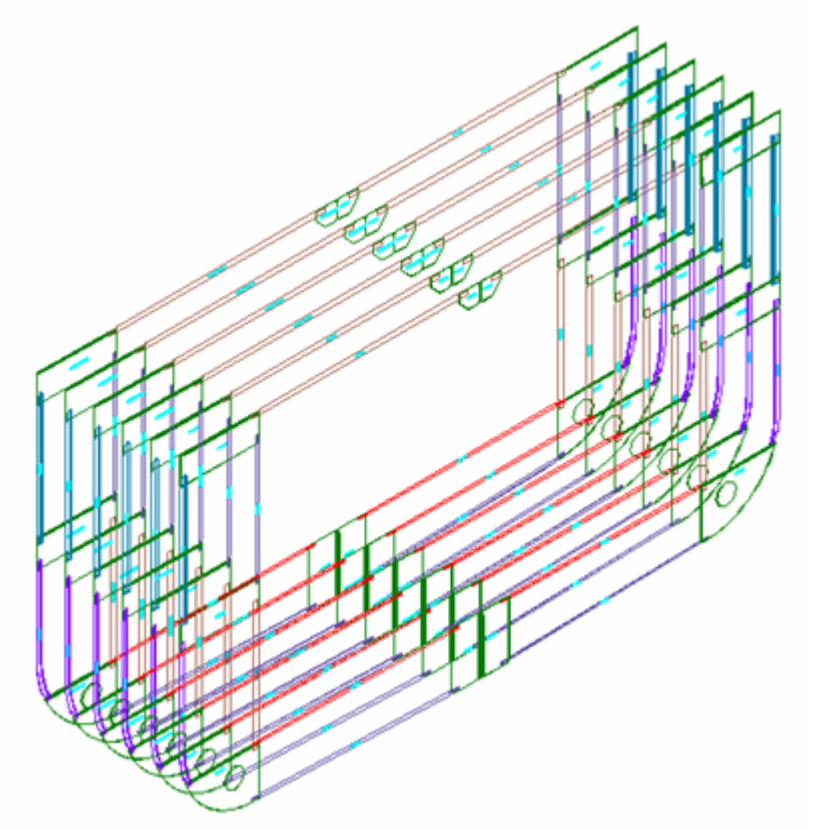
Maximize DDROM Effectiveness (Relationship Management)

To maximize the updatability of your 3D Product Data Model within the DDROM, you need to plan the design to use the largest number of relationships possible. This can be done by minimizing the number of actions you have to perform that break these relationships. The best way to minimize these actions is to think ahead and understand where you want to go.

DDROM Example

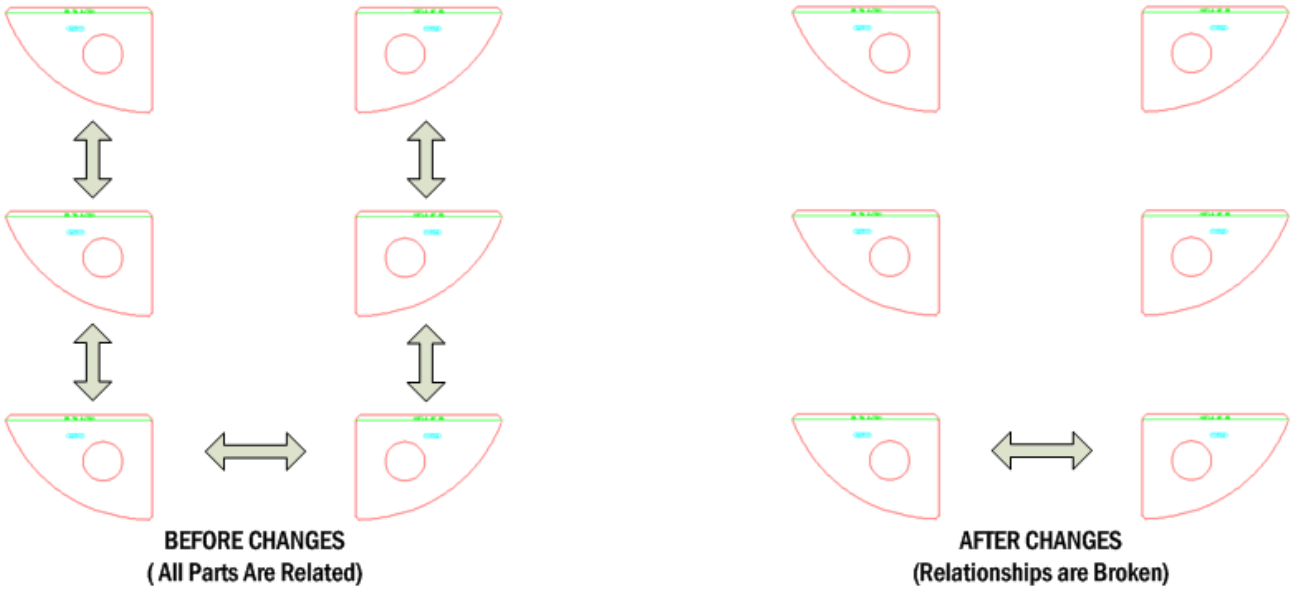
The Goal

The six frames shown in the image below are all largely identical with the exception of minor hull form changes. These changes are significant enough to make the parts on the hull non-identical from frame-to-frame. However, the frames are similar enough that it does make sense to replicate the parts of one frame into the others and make the minimum modifications required.



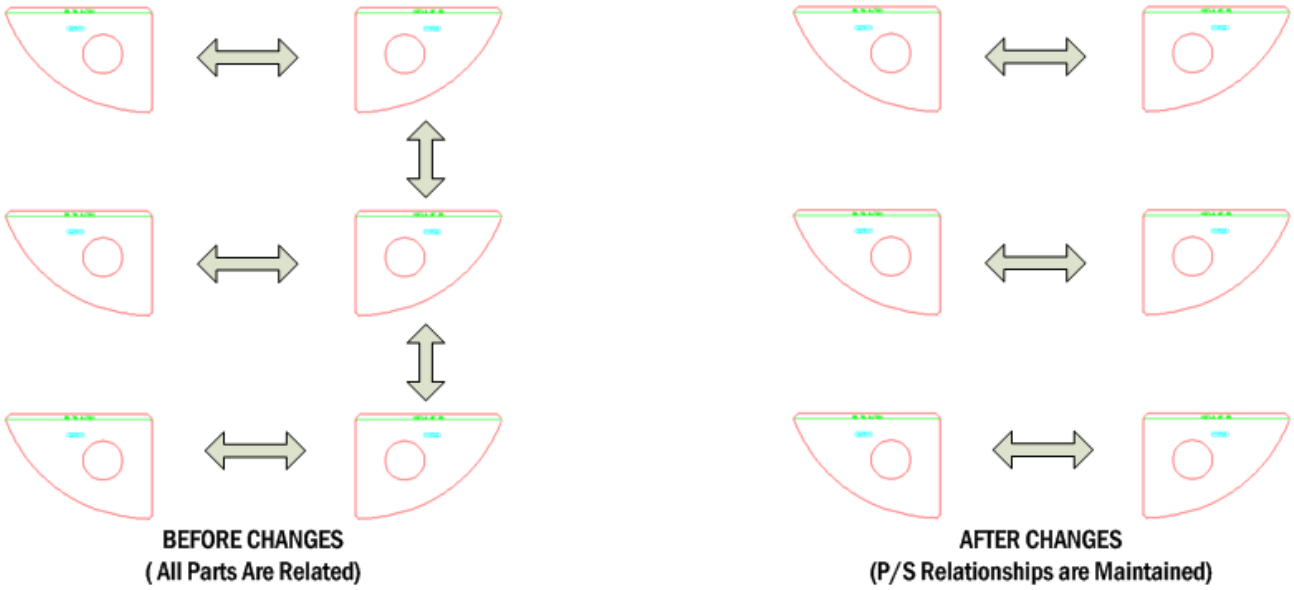
Approach 1

The most obvious approach seems to be to create one side of one frame, mirror all of those parts, and then replicate the entire frame to the other similar frames. This approach has two major failings. The first failing is that only parts that have been explicitly mirrored will be considered mirrored for production purposes (for example, port and starboard nesting). The second failing is caused by the method in which the resulting parts are related to each other. More importantly, as shown in the following diagram, the failure is due to the way the parts are related to each other after the hull form changes are made and the replicated parts are no longer identical, breaking all identical definitions you created.



Approach 2

Because the intention is that parts in each frame maintain a primary mirrored relationship with its port and starboard counterpart, even when we break the identical nature across frames, we want to preserve the relationship across frames to allow it to work within the DDROM. To achieve this result, we can create one side of the first frame, replicate those parts to the similar frames, and then perform the mirror in each frame. As you see from the image below, a little more thought and effort when modeling will make the results more effective in the long run.



Construction Line Management

One of the key elements to ShipConstructor is construction line management. A construction line is a polyline entity that is stored in the database and can be used to define the boundary portions of a ShipConstructor part. In a large project that takes advantage of the DDROM to its full extent, you will have a large number of construction lines and managing those lines will be key in understanding how your model works.

There are four major types of construction lines: User construction lines, PlanarGroupPlane construction lines, HullTrace construction lines, and Offset construction lines. These construction lines form the basis for the DDROM. How you create and modify them will impact the complexity and the ease of updating your model.

See Construction Lines (page 151) for more details and procedures on construction lines.

Note: In previous versions of ShipConstructor, you had to trim, extend, break, and perform other procedures on draft lines to achieve one perfect toolpath for a single part. In ShipConstructor, performing too many operations on construction lines can inhibit your ability to make changes later on. Not only does trimming a construction line limit how usable it is in other parts, it will also affect how usable the line is in the part that it is being trimmed for. For example, a horizontal construction line in a frame that is trimmed to the HullTrace construction line for that frame may no longer intersect the HullTrace if the hull form is changed significantly in that area. If it is left as a longer line, the line will be less likely to require user modification when changes happen or when it is replicated to wider frames where it may not otherwise intersect the new HullTrace. This concept applies in many areas.

General Use Tips

ShipConstructor introduces us to many new concepts and has many new features that should make it easier to create, edit, and manage complex 3D Product Data Models than ever before. Here are some simple tips to help you get started:

- You can define planar groups from any drawing. Keeping all section lines in a single read-only ControlLines.dwg will make managing those sections much easier in the long term.
- While you are still growing accustomed to ShipConstructor and the DDROM, pay attention to the relationships newly created parts and construction lines have to other items in your model. Relationships for parts and construction lines can be viewed via SC Structure > Parts > Information... (page 433) or SC Structure > Construction Line > Information... (page 439), respectively. This is also where relationships can be broken if curves are to be added or removed.
- Plan ahead. Think about the model you are working toward rather than just the part you are trying to create.
- Part naming and updating part names has been greatly improved. Use naming conventions whenever possible and do not edit names manually unless you are sure you need to. (If you edit a name manually, the name is no longer a generated name and therefore will not be updated to reflect later changes such as a change in product hierarchy assignment.)
- Save drawings often. Auto-save does not save to the ShipConstructor database, and any conflicts with other users of the DDROM will be uncovered on a save.

Structural Modeling

Structural modeling in ShipConstructor is very different from previous versions of ShipConstructor. This document provides an overview of the new structural modeling workflow and highlights some of the improvements.

Database

ShipConstructor now stores all geometry, parts, and so on, and their attributes (for example, material, weight, revisions, product hierarchies) within the project database. Whenever you open a drawing, ShipConstructor regenerates the drawing using the information in the database. Previously, ShipConstructor stored geometry within the drawings.

In ShipConstructor, any changes you make to a drawing are not saved to the database until you save the drawing. In ShipConstructor2005, the database is updated as you work in a structural drawing. If two users are editing different versions of a common (identical or mirrored) part, the first user to save their drawing has their edits saved to the database; the second user is notified of a concurrency issue.

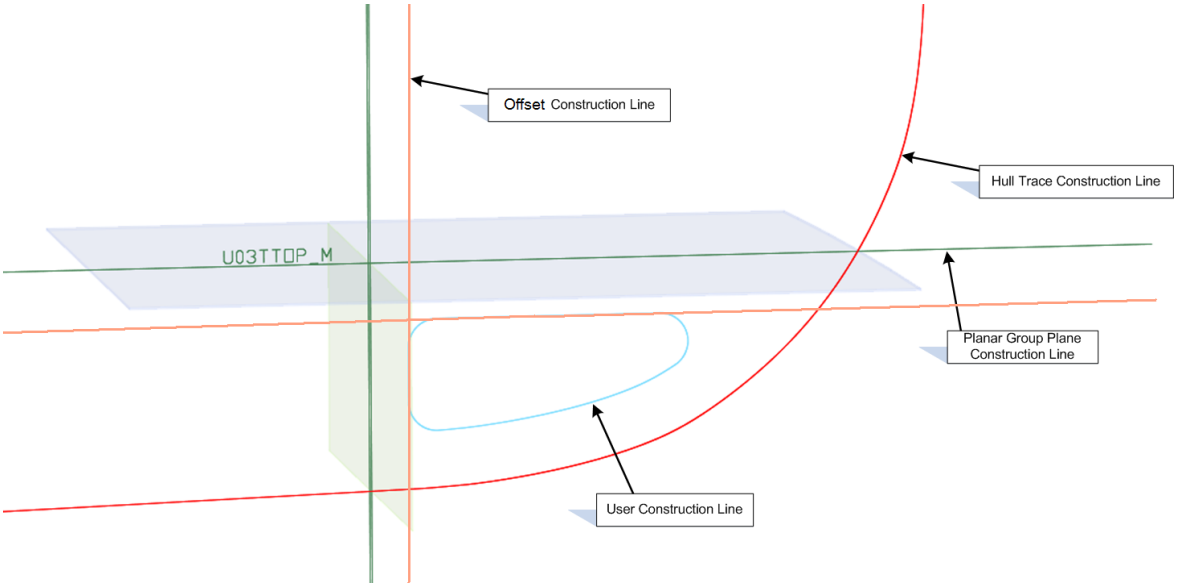
3D Unit Drawings

As in ShipConstructor2005, the purpose of the 3D unit drawing is to provide a 3D environment for you to define the geometry of planar groups and hull traces. You can define hull trace geometry using the Hull module, using ShipCAM, or using AutoCAD drawings or geometry.

Construction Lines

Construction lines are a new entity in ShipConstructor. They are drafting lines that are used to define and edit the boundaries of each part. These drafting lines are meant to be for developing the production parts.

There are four types of construction lines: HullTrace construction lines, PlanarGroupPlane construction lines, Offset construction lines, and User construction lines.

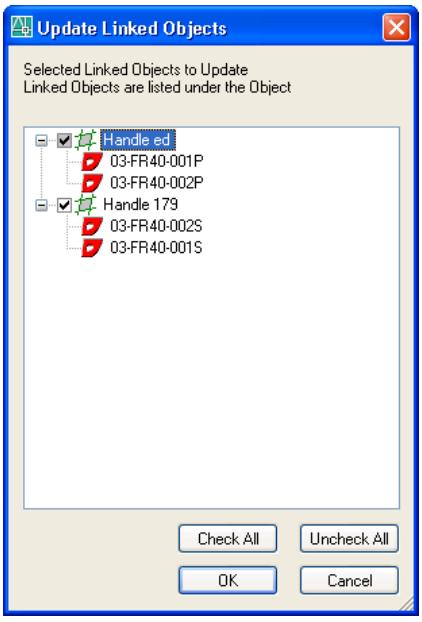


Note: In ShipConstructor2005 you add details (for example, scallops, cutouts) to part boundaries before defining the part. In ShipConstructor you add details after defining the part.

Modifying Construction Lines

Whenever you modify a construction line, ShipConstructor automatically updates all associated parts.

Construction lines can be linked to another construction line, making it easy for you to modify one construction line and have all the changes applied to linked construction lines. When modifying a construction line by moving, rotating, extending, and so on, ShipConstructor will display the Update Linked Objects window.



The construction line with object handle ed is being modified. The construction line has two plate parts that use the construction line for definition 03-F40-001P and 03-F40-002P, and one identical construction line Handle 179 (which also has two plate parts using the construction line for definition 03-FR40-001S and 03-FR40-002S).

The Update Linked Objects window displays all the linked construction lines and all the parts that use those construction lines. This window lets you select which linked construction lines you want to apply the current modification to. For any of the linked construction lines that are checked, the current modification are applied to them and the parts that use them. For any of the linked construction lines that are not checked, no changes are applied to them, and the link between the construction lines are broken.

HullTrace Construction Line

Very few modifications can be done to a HullTrace construction line. Once a hull trace is created you will not be able to change the shape of the hull trace the way you can for User construction lines. You are able to trim, break, and extend the HullTrace construction line but are not able to move, rotate, and edit grips. You should limit the amount of trimming and extending done to the HullTrace construction line, as explained in the Construction Line Management (page 150) section.

You can replace the geometry of a HullTrace construction line with the geometry of another line.

To replace a hull trace construction line's geometry:

1. Define a new line or locate an existing polyline, line, or construction line in the planar group of the hull trace construction line.
2. Choose SC Structure > Construction Line > Replace Construction Line Geometry (page 442)

   ```
   Select original construction line:
   ```
3. Select the original hull trace contruction line.

   ```
   Select new line geometry:
   ```
4. Select the line with the desired geometry.

ShipConstructor automatically updates all parts associated with the original hull trace. (See Construction Line Relationships (page 154).)

PlanarGroupPlane Construction Line

PlanarGroupPlane construction lines represent the intersection of two planar group planes. (A planar group plane is like a UCS, except that it resides in the project database. A UCS only resides in drawings.) You can create PlanarGroupPlane construction lines using the mark group intersections command.

> Note: This command works similarly to the same command in ShipConstructor2005, except ShipConstructor creates a PlanarGroupPlane construction line instead of an AutoCAD polyline.

The created PlanarGroupPlane construction line is linked to the two planes that created it. If one plane is modified through the move planar group command, any PlanarGroupPlane construction line defined by that plane will be automatically updated to the new intersecting plane. If the planar group plane is used in defining a plate part boundary, the plate part will also be updated automatically.

There are two ways you can create PlanarGroupPlane construction lines:

- ShipConstructor automatically creates PlanarGroupPlane construction lines within a planar group drawing when you mark group intersections. (That is, to create PlanarGroupPlane construction lines within a planar group drawing, simply mark group intersections.)
- ShipConstructor automatically creates a PlanarGroupPlane construction line within a planar group drawing when you activate a UCS using SC Utilities > Activate UCS (page 451). (ShipConstructor does not automatically create a PlanarGroupPlane construction line when you activate a UCS using AutoCAD's **UCS** command.)

Each PlanarGroupPlane construction line is directly related to a planar group, so the only way to modify a PlanarGroupPlane construction line is to modify (for example, move) the entire parent planar group. ShipConstructor will then automatically update all parts associated with the PlanarGroupPlane construction line.

To move PlanarGroupPlane construction lines

1. Choose SC Structure > Move Planar Group (page 449).
2. Enter the amount to move the planar group and the direction to move it in.
3. ShipConstructor displays a window listing other UCSs that you can choose to move. Select the UCSs that you want to move.

ShipConstructor automatically updates all parts associated with the PlanarGroupPlane construction line and any construction line that is linked to those PlanarGroupPlane construction lines.

Offset Construction Lines

Offset construction lines are created from other construction lines and will maintain the offset relationship with the original line. They are created with the AutoCAD **OFFSET** command. If the line selected for the command is a construction line, then a new offset construction line will be created. Offsets can be created from any type of construction line, including other offset construction lines. Modifications that change the shape of the line are not allowed on offset construction lines, but it may be moved, rotated, trimmed, or extended (see below). The offset construction line maintains the relationship with the source construction line, similar to the relationship between identical construction lines, and changes to one will affect the other (see Construction Line Relationships (page 154)).

The offset value of an offset construction line can be changed with the SC Structure > Construction Line > Edit Offset (page 441) command.

Offset construction lines can be trimmed, extended, or broken into multiple offset construction lines with the AutoCAD TRIM, EXTEND, or BREAK commands. These trims operate similar to other AutoCAD polylines; however, when an offset construction line is trimmed it will always remember its original geometry. This means, for example, that if a trimmed offset construction line is extended, it will extend back along the original curve of the line, maintaining its original geometry and relationship with its source construction line.

Offset construction line trims can be removed with the SC Structure > Construction Line > Clear Trims (page 441) command. This removes all trims and extensions leaving the original offset construction line.

Trims and extensions can also be applied with the offset construction line's grip points. The end points can be dragged along the construction line path to easily trim or extend the line on the fly.

User Construction Lines

User construction lines are the most flexible construction lines. Any AutoCAD command that you can perform on an AutoCAD polyline, you can also perform on a user construction line. These lines represent arbitrary (user-defined) part boundaries. Within the planar group drawing, use AutoCAD's drawing tools to create lines, polyline, circles, and so on that represent the boundaries of parts. When you create a part that uses these lines as boundaries, ShipConstructor automatically converts the lines to User construction lines. When you move or modify User construction lines, ShipConstructor automatically updates all associated parts.

There are several ways that a User construction line can be created.

To convert an AutoCAD line to a construction line

1. Choose SC Structure > Construction Line > Convert to User Construction Line (page 441)
2. Select a curve to convert to a user construction line.

Note: Construction lines are made from polylines only, so circles will not remain as circles.

There are other ways to add construction lines to a drawing:

- When creating a plate part or profile part, all used geometry that is not already a construction line will be converted into User construction lines.
- Copying or mirroring of a PlanarGroupPlane construction line, HullTrace construction line, or User construction line will create a new User construction line. Copying or mirroring an Offset construction line will create another Offset construction line. The newly created construction line is linked to the original construction line, so if the original construction line is modified, the new construction line will also be modified.
- When a structure part (plate, stiffener, and faceplate) is copied or mirrored, all the geometry is automatically copied in the form of user or offset construction lines, depending on the source. Again, these construction lines are linked to the original construction lines and will be modified when the original line is modified.

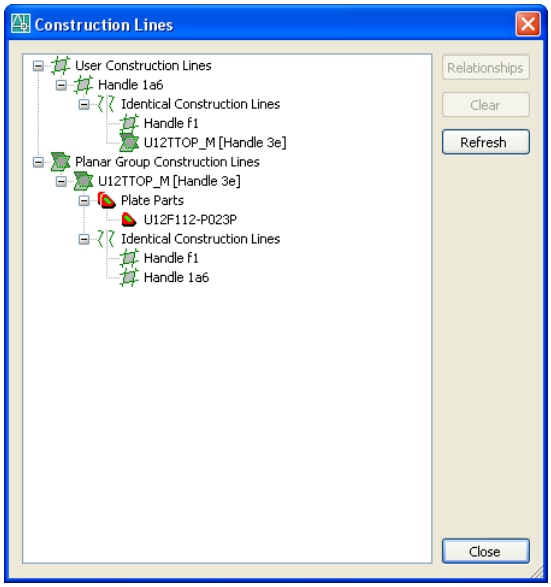
Note: You can restore most construction lines back into AutoCAD polylines by exploding them. You will not be able to explode construction lines associated with parts.

Construction Line Relationships

Construction lines are used to define parts and can be linked to other construction lines. ShipConstructor has a feature to show you all the information that relates to a construction line.

1. Choose SC Structure > Construction Line > Information... (page 439)

2. Select one or multiple construction lines.



User construction line with Handle 1a6 has an identical User construction line (Handle f1) and an identical PlanarGroupPlane construction line (U12TTOP_M [Handle 3e]). The PlanarGroupPlane construction line is used in the definition of plate part U12F112-PO23P.

To replace a construction line

1. Choose SC Structure > Construction Line > Swap Construction Line (page 441)
2. Select the construction line that you want to replace.
3. Select the new construction line.

Structure Part Relationships

In ShipConstructor, parts can have relationships to other parts. This concept is not entirely new since in ShipConstructor2005 you had the capability of having PS parts and identical parts. In ShipConstructor2005 the PS part did not have an object associated with it during modeling until you broke the link. In ShipConstructor, it is possible for parts to have a mirrored relationship (similar to PS parts in ShipConstructor2005) and also an identical part. In ShipConstructor all parts have an object associated with them.

Displaying Part Relationships

1. Choose SC Structure > Parts > Relationships... (page 432).
2. Select a ShipConstructor structural part for display.

The Relationships window displays all the relationships that you need to know about the part.

How to Create Related Parts

There are two types of relationships between parts. Parts can be related as mirrors or identicals.

To create mirrored parts

You can create mirrored parts using one of two methods:

1. Use the AutoCAD **MIRROR** command.
2. Use the ShipConstructor command SCMIRRORCL.

After either of these two commands, the source and the destination part will be linked as mirrored parts. All needed construction lines and attached parts (stiffeners, faceplates) are also automatically mirrored. These parts also have the mirrored parts link with each other.

To create identical parts

1. Use the AutoCAD **COPY** or **ARRAY** command.
2. Choose SC Structure > Replicate Objects to Other Group (page 449)

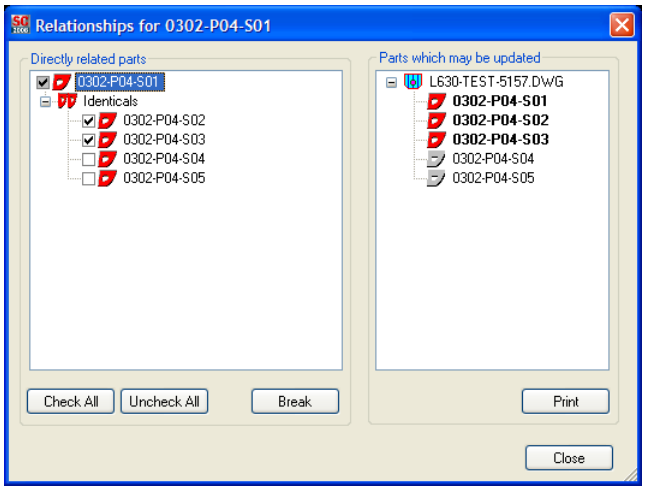
The source and the destination part will be linked as identical parts. All needed construction lines and attached parts (stiffeners, faceplates) are also automatically copied. These parts are also identical to their original parts.

Removing Links Between Parts

Through the Parts Information window, you can remove links between parts. This window provides a link to the Relationships window for the selected part. Optionally, it is possible to launch the Relationships window for the part directly. See Displaying Part Relationships (page 155).

To remove the link between parts:

1. Choose SC Structure > Parts > Information... (page 433)
2. Select the part you want to remove links for.
3. Select the part in the Parts Information window.
4. The Relationships button is now enabled. Click it to display the Relationships window for the part selected in the previous step.



5. ShipConstructor displays the Relationships window. Uncheck all the parts that you want to break links with. Note that all parts checked will stay linked to each other. For example, in the above image parts 0302-P04-S01, 0302-P04-S02, 0302-P04-S03 will no longer be linked with parts 0302-P04-S04, 0302-P04-S05, but will remain linked with each other.
6. The Break button will now be enabled. Click it to break the links between parts.
7. Click Close when finished editing the relationships and return to the Parts Information window.

Fixing Identical Part Names

There are situations that may cause two or more identical parts to have identical relationships between them but the parts have different names or different name structures. There are two commands to correct the part names of identical parts depending on the results desired. Both these commands will correctly handle parts in different assemblies and with different part sides as long as your part naming conventions are set up to do so. Both commands should only be run in planar group model drawings and will try and fix the names of identical parts that reside in the current drawing as well as in other drawings. Multiple parts (even of different types) can be selected at once.

To fix identical part names using the name of a selected part:

1. Open a planar group model drawing and select the structure parts with the names you wish to use as a basis for the names of the parts with identical relationships to them.
2. Run the command SCFIXIDENTPARTNAMES

This will use the part name of selected parts (generated or non-generated) as the basis for the names of all parts with identical relationships to them.

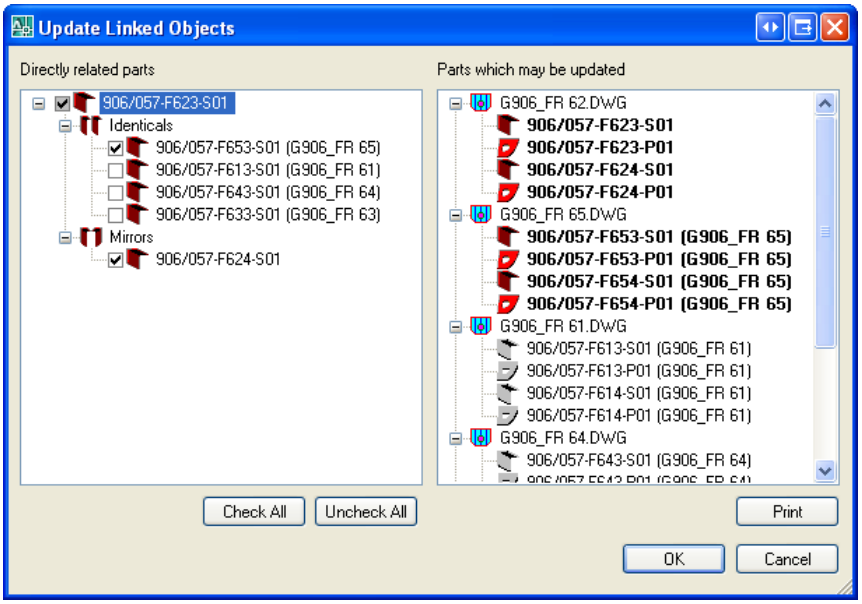
To fix identical part names using the lowest auto-number:

1. Open a planar group model drawing and select the structure parts that you wish to fix the names of.
2. Run the command SCGENERATEIDENTPARTNAMES

This will find the generated part name with the lowest auto-number for each selected part and their identicals and use this auto-number to regenerate the part names of all the parts with an identical relationship to the selected parts. Parts that have non-generated names will be changed to have generated names.

Updating Mirrored and Identical Parts

When you change or edit parts that are related to other parts the Update Linked Objects window appears, giving you a choice of which related parts to update.



The left side of the window shows the parts that are directly related via an identical or mirrored relationship to the original part you edited or changed. Checking the box beside each part selects that part to be updated with the changes you made to the original part. If the box beside a part is not checked, that part will not be updated with any changes and the relationship between it and the original part will be removed. The right side of the window shows a list of parts and the drawings they are in that will be updated or affected by the changes you are making to the checked parts. For any part listed that resides in the current drawing you can right-click on it and select Zoom to zoom the current view to that part. The relationship of any part listed on the right side of the window to the original part being changed can be seen by right-clicking on the part and selecting Find Related Path.

Note: All parts that are checked will remain related to each other but not to the unchecked parts. Likewise, the unchecked parts will all remain related to the other unchecked parts, but not to the checked parts.

Mark Group Intersections

When creating or detailing a structural part, you typically need information about other structural objects that do or might intersect the plane of the part. You can find this information by marking, in the current drawing, the intersection of various UCSs (corresponding to other structural parts) with the current UCS.

For example, if a frame is intersected by decks and girders, ShipConstructor calculates the intersection lines between the frame and the decks and girders by calculating the intersection of their UCSs in 3D space. The UCS can either be the construction or thickness UCS or a specially created UCS from other planar group drawings.

To mark group intersections

1. Choose SC Structure > Mark Group Intersections (page 393) to open the Mark Group Intersections Window (page 393).

   ShipConstructor inserts the marks as follows. All lines are ten percent longer than the current extents of the drawing. You will have to trim the lines later to their appropriate sizes using standard AutoCAD trimming.

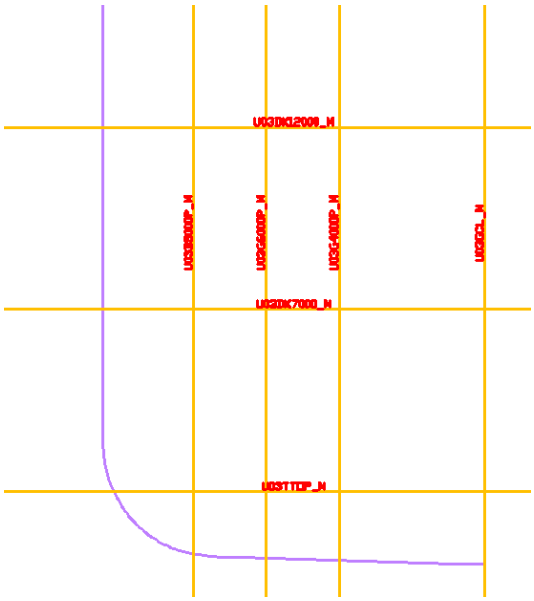


Plate Parts

A plate part is a part made from plate stock. Plate parts are generally flat. Plate parts can also be flanged (see Plate Flanges (page 163)).

Plate Part Creation

ShipConstructor uses a new process for creating plate parts. (It is still possible to use the ShipConstructor2005 process for creating plate parts in ShipConstructor.)

The following improvements have been made to plate parts:

- Plate parts are now parametrically defined from construction lines. Moving a construction line will automatically modify the part associated with the construction line.
- You can now modify or remove holes, cutouts, scallops, and other elements without having to redefine the part. This change makes model revisions much faster.
- Cutouts are now associated with stiffeners. When you change a stiffener, the cutout in the plate is automatically changed.

  Note: Replicated geometry may not be identical to the original geometry because stiffeners and cutouts may not be in identical places.

- Piecemark and orientation icons are now placed automatically, reducing drafting time.
- There is a new interface for creating plate parts.

Previously, you assigned a part to the product hierarchy during part creation. Now, you assign the part to the product hierarchy when editing the part.

You can now define the piecemark location by selecting the inside boundary point.

If the plate part will have holes, then you need one closed toolpath on the _Draft_Cut layer for each hole.

To create a plate part

Note: Unlike ShipConstructor2005, you do not need to define a closed toolpath and extrude a solid before defining a plate part.

1. Define the boundaries of the part using HullTrace construction lines, PlanarGroupPlane construction lines, Offset construction lines, User construction lines, and polylines. Use closed polylines to define inner boundaries (holes).
2. Choose SC Structure > Plate > New (page 397)
3. Click within the part boundaries.

   The point that you click will become the location of the part's piecemark.

   ShipConstructor automatically detects the part boundaries.

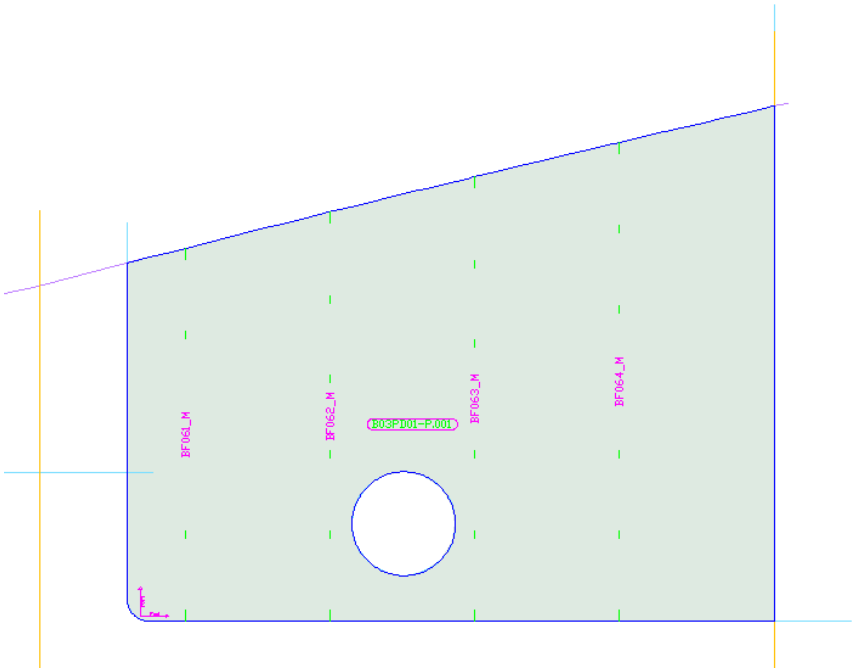
   The Edit Properties window appears.

To edit a plate part

1. Choose SC Structure > Plate > Edit (page 400) and select the plate part or right-click on the plate part and choose Edit Properties.

   The Edit Properties window appears.
2. Set the desired properties.
3. Click OK.

   Note: Changes made to the Part name, Product Hierarchy, Finishes and User Attributes are not applied to the mirrored and identical parts at this time.



A plate part (shaded)

Copy a Plate

To copy a plate

A plate part can be copied by using the AutoCAD **COPY** command.

Note: All attached extrusions and boundary construction lines will be copied with the part. The plate, attached extrusions and the boundary construction lines will have an identical relationship with their source.

Mirror a Plate

To mirror a plate

A plate part can be mirrored by the following methods:

- Using the AutoCAD **MIRROR** command.
- Using SC Utilities > Mirror about Centerline (page 461)

Note: All attached extrusions and boundary construction lines will also be mirrored with the part. The plate, attached extrusions and the boundary construction lines will have a mirror relationship with their source.

Move a Plate

To move a plate

A plate part can be moved by using the AutoCAD **MOVE** command.

Note: You cannot use the mouse to drag a plate part. This operation is disabled within planar group model drawings.

If the plate you are moving has any mirrored or identical parts, you are asked which of these parts you want to be moved. Mirrored parts are moved relative to their mirrored axis. See Updating Mirrored and Identical Parts (page 157).

Detail a Plate

Detailing a plate part can include inserting scallops or cutouts or adding marking.

In ShipConstructor2005 you insert details (scallops, cutouts, and so on) before defining the part. In ShipConstructor you can insert details after defining the part. When you detail a part, ShipConstructor automatically updates the part and its associated construction lines.

Add, Remove, and Edit Objects on an Existing Plate Part

You may forget to include a marking line or similar object when defining a plate part. You can add objects later by using the Add Objects command. For each part that you run the command on, you will be asked which objects you want to add. You can only add objects to a plate part. You can only add production objects to a part.

There are several types of objects that can be added to a plate part.

- Text objects
- Marklines (Use markline style defined in Manager > Structure > Plates > Marking Styles > Assign Styles under the General Assigned Styles)
- Cut lines (if curve is on _Draft_Cut layer, otherwise cut lines are added as marklines)

To add an object to a plate part

1. Choose SC Structure > Plate > Add Objects (page 404)
2. Select all plate parts that you want to add an object to.
3. Select the object to add to all selected plate parts.

   The selected objects are added to the existing part.

To remove an object from a plate part

1. Choose SC Structure > Plate > Remove Objects (page 406)
2. Select the object on the plate part that you want to remove.

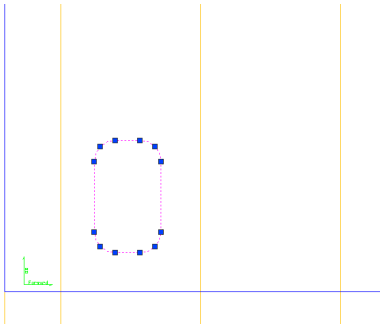
To edit objects of a plate part

You can modify text, marklines, piecemarks, the part orientation icon, and weld shrinkage properties of a plate part.

1. Choose SC Structure > Parts > Edit Component (page 435)
2. Select on the component of the plate part that you want to edit. Depending on your selection an edit window will appear allowing you to change the settings for that instance of that component.

Add a Manhole to a Plate

1. Make the _Draft_Cut layer the current layer.
2. Draft a manhole as a polyline using regular CAD drafting commands. Ensure the polyline is closed.



3. Choose SC Structure > Plate > Add Objects (page 404).
4. Select the plate part and press Enter.
5. Select the manhole polyline and press Enter.

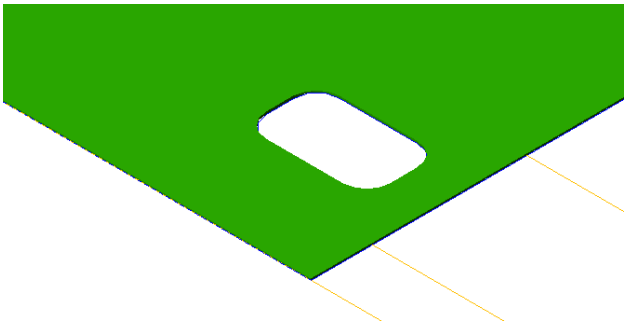


Plate Corner Treatments

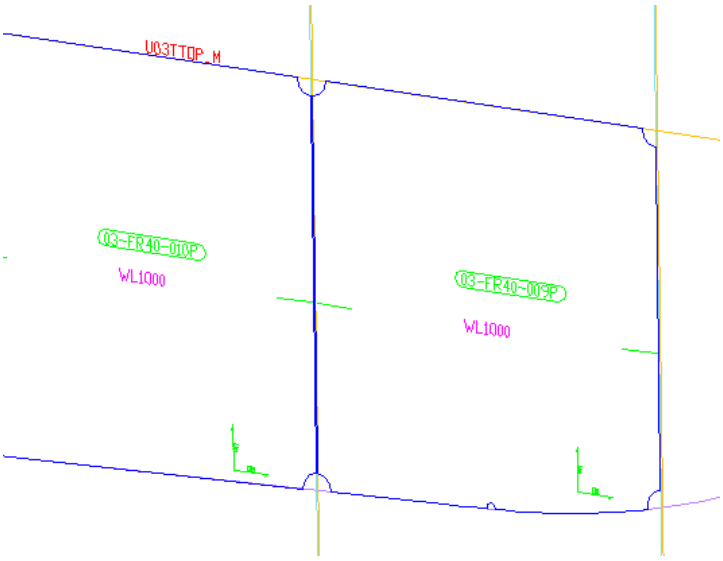
In ShipConstructor, you create corner treatments on plate parts after you have defined a plate part. The different types of corner treatments supported in ShipConstructor are snipes, fillets, and scallops. Before you can add corner treatments, you need to make a corner treatment standard in Structure > Plates > Corner Treatments (page 355).

To add a corner treatment

1. Choose SC Structure > Plate > Corner Treatment > Insert (page 407).

   Select Plate parts:

2. Select all the plate parts that you want to add corner treatments to. Press Enter.
3. Select the corner treatment that you want to add to the selected plate parts.
4. If you have Zoom to part checked, ShipConstructor will zoom to the first plate part that you selected. Select the points that you would like to add the corner treatment to.
5. Press Enter when you are done.
6. ShipConstructor will cycle through the plate parts that you selected to add corner treatments to. Repeat Steps 4 and 5 until all corner treatments have been added.



Note: You can also use corner treatments to add weld seam reliefs to plate parts.

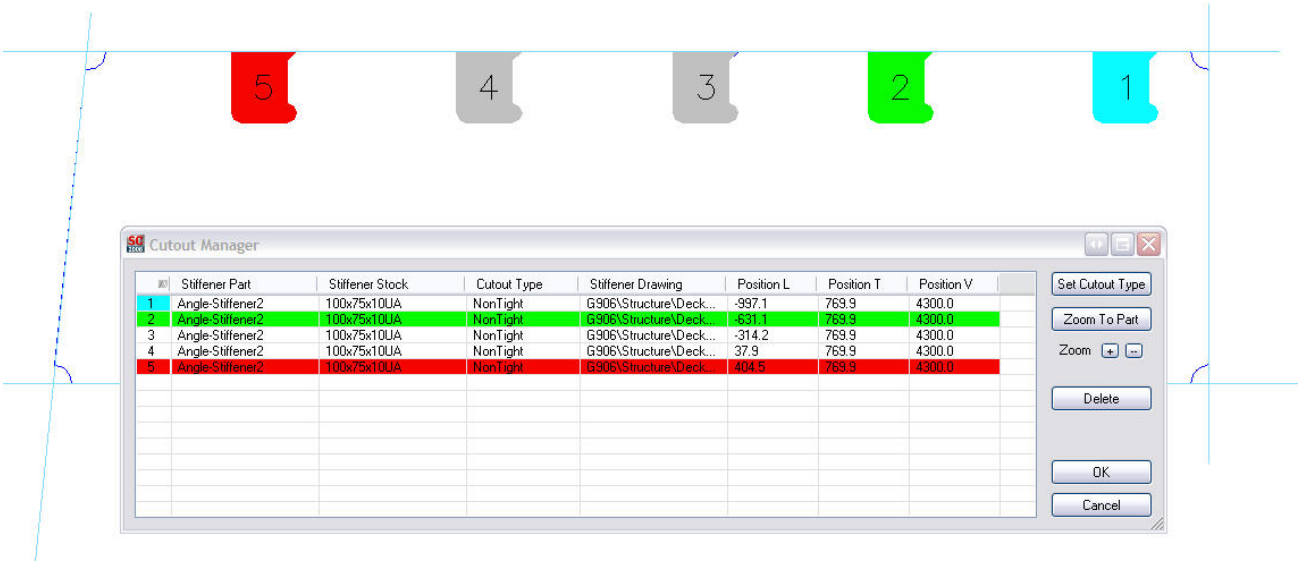
Plate Cutouts

Stiffener cutouts are also added to the plate part after the part is defined. The part solid and production information is automatically updated. Before a profile can create a cutout through a plate part, the stock of the profile must have a cutout shape associated with the stock. Refer to the Create a Cutout (page 67) section to do this.

To add a cutout

1. Choose SC Structure > Plate > Manage Cutouts... (page 406)
2. Select a plate part.

   ShipConstructor determines all the stiffeners that intersect the plate and displays them in the Cutout Manager.


5
4
3
2
1
Cutout Manager
Stiffener Part | Stiffener Stock | Cutout Type | Stiffener Drawing | Position L | Position T | Position V
1 Angle-Stiffener2 100x75x10UA NonTight G906\Structure\Deck... -997.1 769.9 4300.0
2 Angle-Stiffener2 100x75x10UA NonTight G906\Structure\Deck... -631.1 769.9 4300.0
3 Angle-Stiffener2 100x75x10UA NonTight G906\Structure\Deck... -314.2 769.9 4300.0
4 Angle-Stiffener2 100x75x10UA NonTight G906\Structure\Deck... 37.9 769.9 4300.0
5 Angle-Stiffener2 100x75x10UA NonTight G906\Structure\Deck... 404.5 769.9 4300.0
Set Cutout Type
Zoom To Part
Zoom
Delete
OK
Cancel

The Cutout Manager displays all the cutouts for the selected plate part. Through the Cutout Manager, you can delete cutouts and edit the cutout type (using the drop-down menu in Cutout Type column). Cutouts are color coded as follows:

Green – New

Red – Deleted

Yellow – Modified

Cyan - Selected

Note: Cutouts for stiffeners that belong to a different unit are added according to the **Allow Remote Profile Cutouts** option, found in the **Structure** section of the project settings (General > Project Settings (page 327)). If this is set to 'No', these cutouts will show up as deleted.

Note: If a stiffener is moved, ShipConstructor automatically updates all cutouts that the stiffener creates.

Note: If a profile stock changes, ShipConstructor updates all cutouts using the new stock's defined cutout and tries to apply the same cutout type.

Plate Flanges

You can add flanges to plate parts. Adding a flange to a plate first requires a flange standard to be created. See Create a Flange Standard (page 79). A flange can only be applied to a straight segment of a plate part. See Flange Catalog (page 79) for more information on flanges.

To add a flange to a plate

1. Choose SC Structure > Plate > Add Flange (page 401).

   ```
   Select edge on plate:
   ```

2. Click on the edge of the plate on the segment that you want to flange. It is best to pick near the middle of the edge because later modification of construction lines could cause the flange to move to a different edge.

   The Select Flange window appears.

3. Select the flange standard, the direction to flange, and the angle to bend the plate.
4. Click OK.

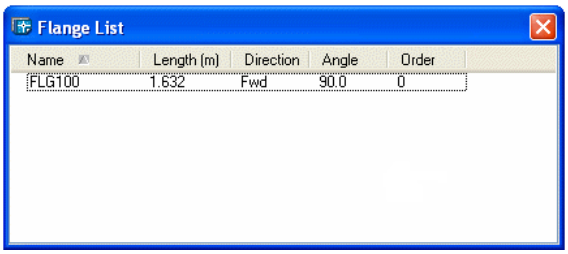
To edit a plate flange

1. Choose SC Structure > Plate > Edit Flange (page 402)

   ```
   Select Plate:
   ```

2. Click on the plate that contains the flange you want to edit.

   The Flange List window appears.

   This window lists the flanges attached to the specified plate solid.

3. Select the desired flange from the list, right-click, and select Properties.
4. The Select Flange window appears, letting you change the flange.
5. Click OK to save the modified flange.
6. Click (in the upper right corner of the Flange List window) to close the window.

To remove a flange from a plate

1. Choose SC Structure > Plate > Edit Flange (page 402).

   ```
   Select Plate:
   ```

2. Click on the plate that contains the flange you want to edit.

   The Flange List window appears. This window lists the flanges attached to the specified plate solid.

3. Select the desired flange from the list, right-click, and select Delete.
4. Click (in the upper right corner of the Flange List window) to close the window.

Plate Green (or Stock)

In ShipConstructor2005, to simulate adding green stock to a plate you have to modify the part's toolpath after you generated the solid. In ShipConstructor we have green standards that can be added to the plate part. First you need to define green standards in Manager under Structure > Plates > Green Standards. After you define a green standard, you can add green to a plate part.

To add green to a plate part

1. Choose SC Structure > Plate > Add Green (page 403).
2. Select the plate part you want to add green to. If the plate part has identical or mirrored parts then you will see the Update Linked Objects window. Select the parts you want to change.

   ```
   Select the start point:
   ```

3. Select the point on the outer edge of the part where the green will start.

   ```
   Select the end point:
   ```

4. Select the point on the outer edge of the part where the green will end.

   ```
   Select segment for green:
   ```

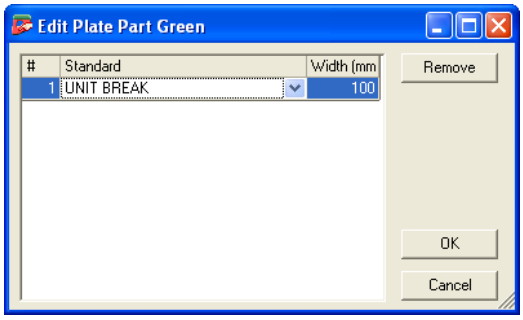
5. Select a point on the segment to add the green to.

6. Click OK.

You can change the standard after you have added green to parts, and the parts are updated when the drawing is opened or when SC Utilities > Reload Drawing is run.

To change green on a plate part

1. Choose SC Structure > Plate > Edit Green (page 403).
2. Select the plate part to edit green. All of the separate green portions are listed for the selected plate part.



The currently selected green in the list is highlighted in cyan in the drawing.

3. From the drop list in the Standard column you can change the standard to a different standard.
4. Click OK.

To remove green from a plate part

1. Choose SC Structure > Plate > Edit Green (page 403).
2. Select the plate part to edit green. All of the separate green portions are listed for the selected plate part.

   The Edit Plate Part Green window appears.
3. Select the Green you want to remove in the list and click Remove.
4. Click OK.

Weld Shrinkage Icon

Weld shrinkage is the amount that a part shrinks when another part (for example, a stiffener) is welded to it. You can compensate for weld shrinkage by inserting a weld shrinkage icon onto a plate part. During nesting, ShipConstructor will oversize the plate part in each direction of the weld shrinkage icon by a specified amount. The weld shrinkage icon is not marked on the plate.

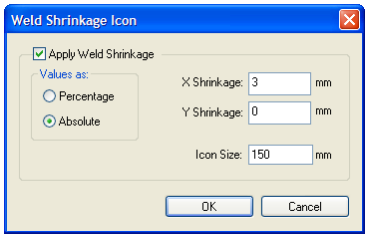
Weld shrinkage depends on the type of material, parameters of welding, the initial temperature of the part (accounting for the differences in summer and winter), and so on. Typically, shipyards use statistical test data from panel lines, data from test welds, or data from previous projects to predict the amount of weld shrinkage.

Note: A good starting point for large vessel steel construction is 1 mm of weld shrinkage per one item of continuous weld.

To insert a weld shrinkage icon

1. Choose SC Structure > Plate > Manage Weld Shrinkage Icon.
2. Select the plate part to add the weld shrinkage to.

   The Weld Shrinkage Icon window appears.



3. Set the options and then click OK.
4. The icon appears on the plate part.
5. Rotate the icon to the desired orientation. The direction of the two axes should be aligned with the stiffener weld directions.

During nesting, ShipConstructor oversizes the plate part in each direction of the weld shrinkage icon by the value given. In the example above, the amount is 6 mm in the horizontal axis and 0 mm in the vertical axis. The icon can be rotated in any direction to accommodate weld shrinkage depending on the direction of the welding.

To remove a weld shrinkage icon

1. Choose SC Structure > Plate > Manage Weld Shrinkage Icon.
2. Select the plate part to remove the weld shrinkage from.
3. The Weld Shrinkage Icon window appears.
4. Uncheck the Apply Weld Shrinkage check box.
5. Click OK.

Dynamic Marking Blocks

Dynamic marking blocks (DMBs) are collections of geometry and text that can be added to a part as a single entity. To add a dynamic marking block to a part use the command SC Structure > Parts > Add Dynamic Marking Block (page 438). For more information see Dynamic Marking Blocks (page 196).

Bevel Information

You can create text objects to represent bevel information that will be used by NC-Pyros to generate bevel cutting NC codes.

You can also create Y-bevels using a single-torch cutting machine to perform multiple-cut passes. Previously, you could only create Y-bevels using a three-torch cutting machine.

To insert bevel information

1. Choose SC Structure > Plate > Bevel > Create (page 409).
2. Select the part that you want to bevel.

```
Select plate part:
```

3. Select a point on the cut path where the bevel starts.

```
Select start point to add bevel to:
```

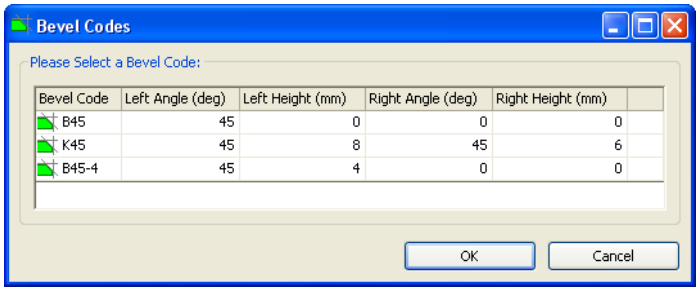
4. Select a point on the cut path where the bevel ends.

```
Select end point to add bevel to:
```

5. Select the correct one of the two edges that you want to bevel.

```
Select red or blue segment to add bevel to:
```

6. The Bevel Codes window appears:



Note: If the bevel is not a symmetrical bevel, you need to determine the correct direction of travel when cutting the part and which side of the part is up when it is nested.

6. Select the bevel code that you want to use and click OK. If the bevel code does not appear in the list, review your bevel standards (see Structure > Plates > Bevel Standards (page 357)).

Automatic Bevels and Plates

If automatic bevels are enabled in the project then bevels will be added to the plate part if necessary. The bevel angles are stored with the construction line if the line is part of a hull surface where the plate contacts the surface at an angle.

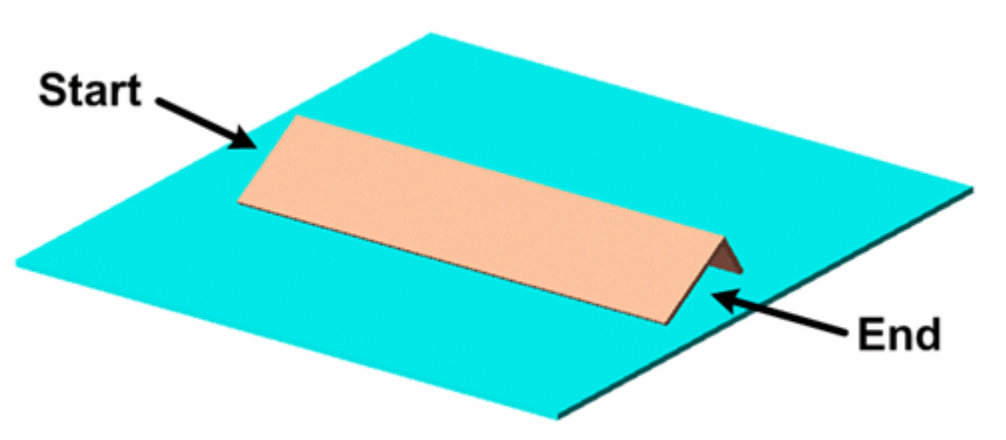
Once automatic bevels are configured and enabled, there is no further action required from the user – the bevels will automatically be added to the plates. Automatic bevels can be configured through Structure > Plates > Bevel Settings (page 357).

Stiffeners

A stiffener is a part made from profile stock or flatbar stock and fastened to a plate part to strengthen it and make it rigid. Most stiffeners are straight, but some stiffeners are curved (for example, for frames built up from profiles).

Rules for stiffener start and end position

ShipConstructor uses the following rule to determine which side of a stiffener is called the start and which side is called the end. Imagine yourself standing in front of a workbench. The stiffener is on the workbench with the length from left to right. The moldline of the web is closer to you, and the flange (or thickness of the web) is lying down toward the workbench surface. The left side of the stiffener is the start; the right side is the end.



Stiffener Creation

To create a stiffener part

1. Create construction lines or AutoCAD curves that define the desired paths of the stiffeners you want to create.
2. Choose SC Structure > Stiffener > New (page 412) to define the stiffener.
3. Select the construction lines or AutoCAD curves to create the stiffeners along and press Enter.

   Note: Only parallel lines can be used to create multiple stiffeners at once. Identical relationships will be created between the newly created stiffeners where possible.

4. If you want to use only a portion of the selected curves to create the stiffeners on, type S at the command line and press Enter. You will be prompted to select two points along each of the selected curves that will be used as the start and end points of the stiffener.

   ```
   Select Plate Parts To Attach to <Enter for None> or [Select stiffener ends]:
   ```

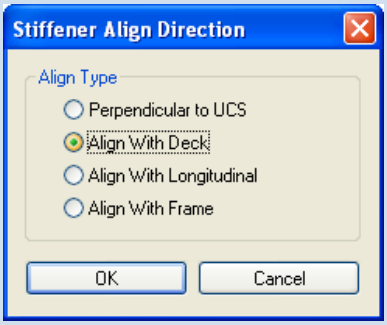
5. If you want to attach the new stiffeners to any plate parts, select the plate parts in the drawing.

```
Select Plate Parts To Attach to <Enter for None> or [Select stiffener ends]:
```

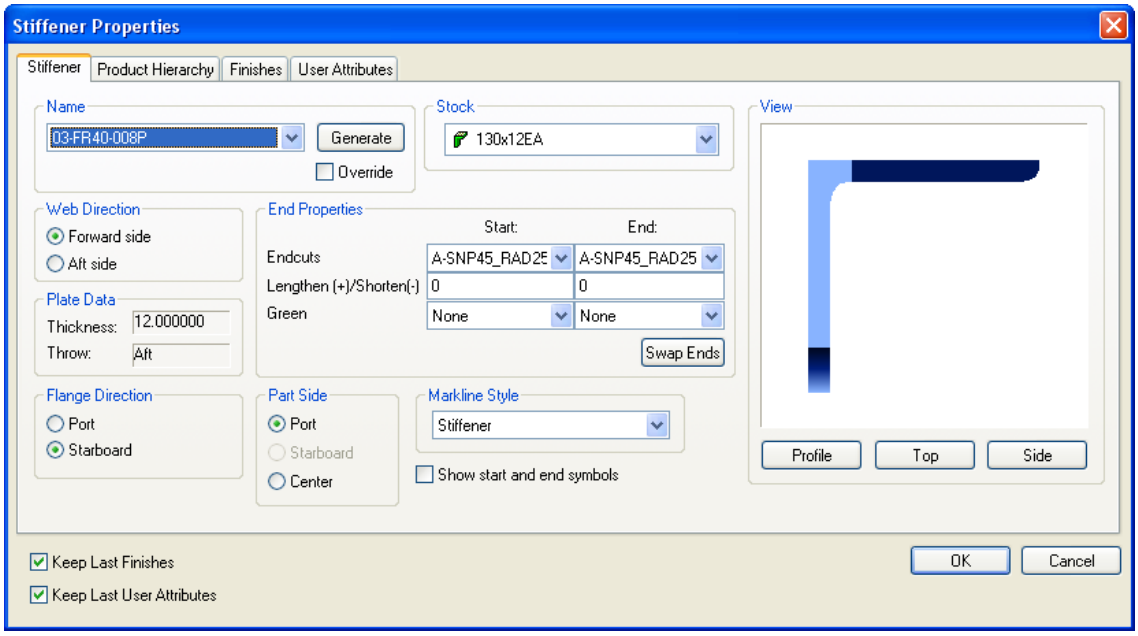
Note: If multiple plate parts are selected and the plate parts have varying thicknesses, then the stiffener will be attached to the surface of the plate with the largest thickness, possibly resulting in a gap between the stiffener and the other selected plates. If none are selected then the stiffener will not be attached to any plate parts.

6. Press Enter.

Note: If a linear entity was selected as the extrude line and the entity's normal is not aligned with one of the axes of the WCS, you will be prompted to select the axis that the new stiffeners will use for their default web direction.



The Stiffener Properties window appears.



7. Set the desired properties. See Stiffener Properties Window (page 412) for details.
8. Click OK.

ShipConstructor does the following:

- Creates the stiffener part including the solid.
- Creates a stiffener markline for each stiffener on any plate parts they are attached to.
- If an AutoCAD entity was selected in Step 1, it will be converted to a user construction line and moved to the _Draft_Mark layer. (You can later move the stiffener by moving the construction line.)
- Calculates bending information to store in the database.

Note: ShipConstructor places the stiffener's piecemark on the stiffener in the plane of the web whereas ShipConstructor2005 places the stiffener piecemark on the plate part.

Note: If the created stiffener is to be attached to a plate part, then the construction line used to create it must be on the same plane as the plate part. This is true even if the stiffener is to be attached to the thickness side of the plate. In this case, ShipConstructor will automatically offset the stiffener by the plate thickness.

To create a stiffener from points

1. Choose SC Structure > Stiffener > New from Points (page 414).
2. Define the path of the stiffener by selecting start and end points for the stiffener. You can keep selecting start and end points to create multiple parallel stiffeners at the same time. Press Enter after you are done defining the paths for all the new stiffeners.
3. If you want to attach the new stiffeners to any plate parts, select the plate parts in the drawing. If none are selected then the stiffener will not be attached to any plate parts. Press Enter.

   ```
   Select Plate Parts To Attach to <Enter for None> or [Select stiffener ends]:
   ```

   The Stiffener Properties window appears.
4. Set the desired properties (see Stiffener Properties Window (page 412) for details).
5. Click OK.

   ShipConstructor does the following:

   - Creates the stiffener parts including the solid.
   - Creates a stiffener markline for each stiffener on each plate part they are attached to.
   - Creates user construction lines on the _Draft_Mark layer for each of the stiffeners. (You can later move the stiffener by moving the construction line.)

   Note: ShipConstructor places the stiffener's piecemark on the stiffener in the plane of the web, whereas ShipConstructor2005 places the stiffener piecemark on the plate part.

Stiffener Edit

To edit a stiffener

1. Choose SC Structure > Stiffener > Edit (page 414) or right-click on the stiffener and choose Edit Properties.

   The Stiffener Properties window appears.

   Set the desired properties (see Stiffener Properties Window (page 412) for details).
2. Click OK.
3. If the stiffener that is being edited has any mirrored or identical parts, you will be asked which of these parts you want to be updated with the changes you made. See Updating Mirrored and Identical Parts (page 157). ShipConstructor does the following:

   - Updates the stiffener and any selected mirrored and identical parts.
   - Updates all marklines and cutouts that reference the updated stiffeners.
   - Re-calculates bending information to store in the database.

   Note: Changes made to the Part name, Product Hierarchy, Finishes and User Attributes are not applied to the mirrored and identical parts at this time.

To edit multiple stiffeners simultaneously

1. Choose SC Structure > Stiffener > Edit (page 414) and select multiple stiffeners when prompted, or pre-select several stiffeners then right-click and choose Edit Properties from the context menu.

   The Stiffener Properties window appears.

Note: Certain properties may not be altered when editing multiple stiffeners so the Stiffener Properties window appears in a limited form. The Product Hierarchy, Finishes and User Attributes tabs are removed and the Name, Web Direction and Flange Direction properties are disabled.

2. Set the desired properties (see Stiffener Properties Window (page 412) for details). Any properties that differ between the selected stiffeners will display “Varies” in the appropriate dialog fields. If any of these fields are changed to a proper value, the updated value will be applied to all selected stiffeners. Leaving a property field as “Varies” (or changing a property and then changing it back to “Varies” before closing the dialog) will leave that property unchanged on selected parts.
3. Click OK.
4. If the stiffeners that are being edited have any mirrored or identical parts, you will be asked which of these parts you want to be updated with the changes you made. See Updating Mirrored and Identical Parts (page 157). ShipConstructor does the following:
   - Updates the stiffener and any selected mirrored and identical parts.
   - Updates all marklines and cutouts that reference the updated stiffeners.
   - Re-calculates bending information to store in the database.

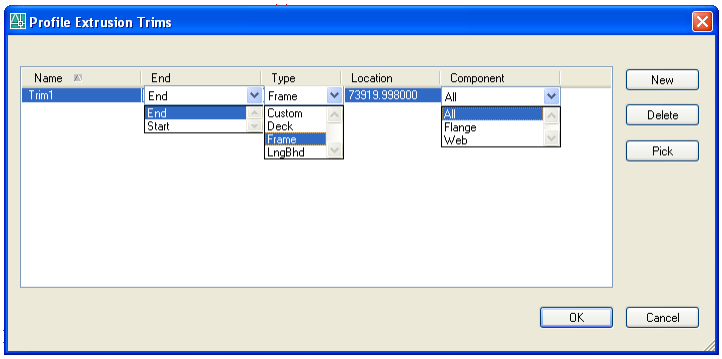
Note: If a stiffener and any of its mirrored or identical parts are selected together when the Stiffener Properties window is opened, the mirrored and identical parts will be automatically removed from the selection set and updated after the window is closed as described in step 4.

Stiffener Trimming

To trim a stiffener (similar for twisted stiffeners and faceplates)

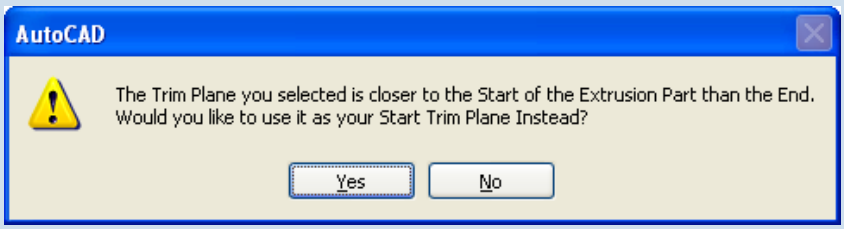
ShipConstructor lets you apply more than one trim to each end of a stiffener, faceplate or twisted stiffener. The trims can also be applied to specific components of the stiffener (for example, trimming just the flange and leaving the web)

1. Choose SC Structure > Stiffener > Trim (page 418) or right-click the stiffener and choose Trim.
2. Select the stiffener you want to trim. The Stiffener Trims window appears.



3. Set the desired options (see Stiffener Trim Window (page 418)).

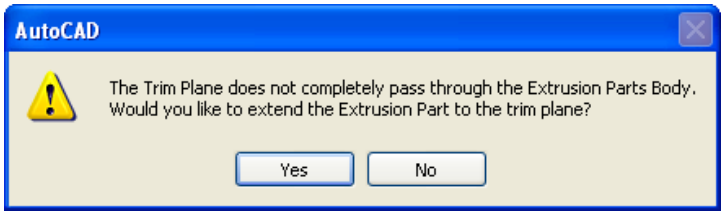
Note: If the selected trim plane is closer to the end that is not currently assigned to the trim being edited, you will be asked if you want to switch the end of the stiffener the trim is applied to.


AutoCAD
The Trim Plane you selected is closer to the Start of the Extrusion Part than the End.
Would you like to use it as your Start Trim Plane Instead?
Yes
No

4. Click OK.

Note: If any of the trims are not valid, an informative message appears describing the error and the Stiffener Trims window is kept open so that you can fix the problem before the trims are saved.

5. If one or more of the selected trim planes does not completely pass through the profile of the stiffener part being trimmed, you will be asked if you want to extend the stiffener to the trim plane. If you choose Yes, the stiffener is extended so that the entire profile of the stiffener intersects the trim plane. If you choose No, only the portion of the stiffener that intersects the trim plane is removed.


AutoCAD
The Trim Plane does not completely pass through the Extrusion Parts Body.
Would you like to extend the Extrusion Part to the trim plane?
Yes
No

6. If the part being trimmed has any mirrored or identical parts, you will be asked which of these parts you want to update with the changes you made. See Updating Mirrored and Identical Parts (page 157).

   ShipConstructor does the following:

   - Updates the stiffener and any selected mirrored and identical parts with the new trim information.
   - Updates all marklines and cutouts that reference the updated stiffeners.
   - Trim angles are calculated to be stored in the database. Please see our online knowledgebase for details on trim angles.

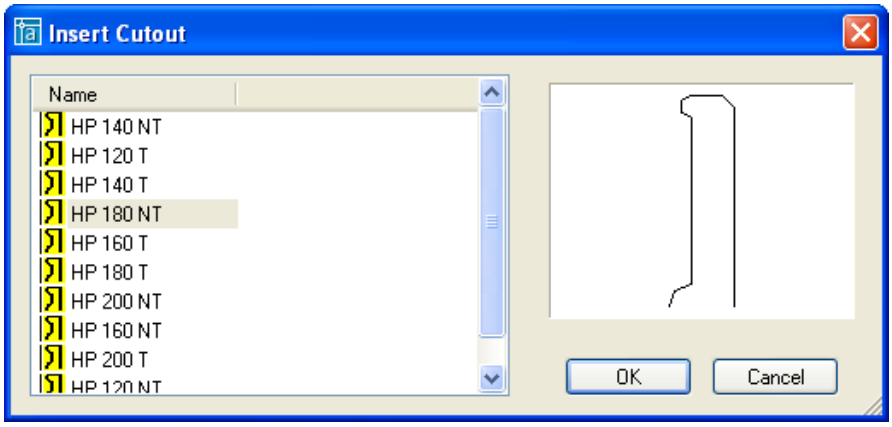
Stiffener Cutouts

To add a cutout to a stiffener (similar for twisted stiffeners and faceplates)

ShipConstructor lets you add cutouts to twisted stiffeners or faceplates as well as stiffeners. The cutouts can only be applied to the web of the selected part.

1. Choose SC Structure > Stiffener > Cutout > Insert (page 416) or right-click on the stiffener and choose Add Cutout.
2. Select the stiffener you want to add cutouts to.

   The Insert Cutout window appears.



3. Select the cutout profile to be used to create the cutouts in the stiffener and click OK. A preview on the right shows the shape of the currently selected cutout.
4. Select the position along the web of the stiffener to insert the cutout. An outline of the cutout shape is displayed at the cursor to aid in placing the cutout. Typing M at the command line and pressing Enter will mirror the cutout profile.

```
Choose Cutout Insert Point<Mirror>:
```

5. Once the insertion point of the cutout is selected, you can modify the orientation of the cutout using the cursor to select the rotation of the cutout profile or using the following command options.
   - OR – Orthogonal
   - T – Tangential
   - P – Perpendicular
   - C – Custom
   - B – Re-pick the cutout position
   - OF – Offset the cutout profile a specified amount along the direction of the web at the cutout insertion point
   - M – Mirror the cutout profile

```
Angle<ORtho/Tangent/Perpendicular/Custom/Back/OFfset/Mirror>:
```

6. Left-click when the cutout profile is in the correct position.
7. Repeat Steps 4 and 5 as many times as needed then press Enter or Esc to finish.
8. If the stiffener that you are adding the cutouts to has any mirrored or identical parts, you will be asked which of these parts you want to update with the changes you made. See Updating Mirrored and Identical Parts (page 157).

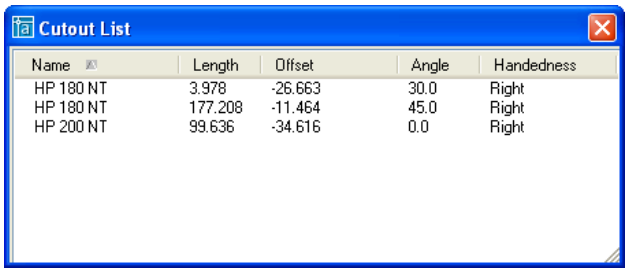
   ShipConstructor adds all the cutouts to the stiffener and any selected mirrored and identical parts.

To edit a cutout on a stiffener (similar for twisted stiffeners and faceplates)

ShipConstructor lets you edit cutouts on twisted stiffeners and faceplates as well as stiffeners.

1. Choose SC Structure > Stiffener > Cutout > Edit (page 417) or right-click on the stiffener and choose Edit Cutout.
2. Select the stiffener whose cutouts you want to edit.

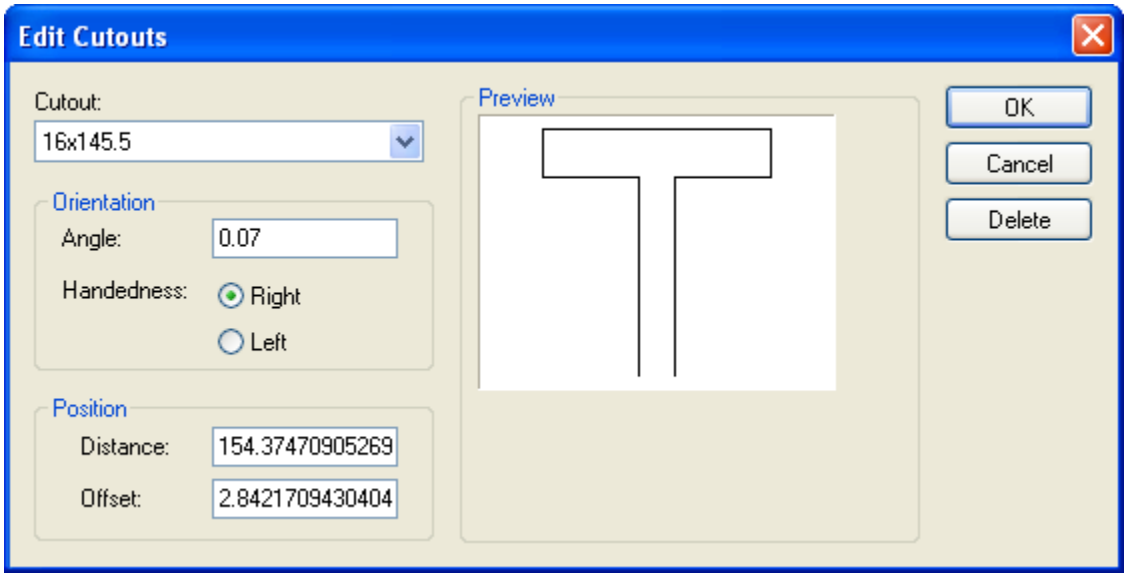
   The Cutout List window appears.



3. Select the cutout to be edited
4. To delete the selected cutout, right-click on it and choose Delete.
5. To move the selected cutout, right-click on it and choose Move. Select a new position for the cutout in the drawing and the cutout will move to that position keeping its original orientation.

   To edit the selected cutout double-click it, right-click on it, and choose Properties or press Enter.

   The Edit Cutouts window appears.



6. Set the desired options (see Edit Cutouts Window (page 417) for details).
7. Click OK.

   The Cutout List window appears.
8. Click Close in the Cutout List window when you are done editing the cutouts.

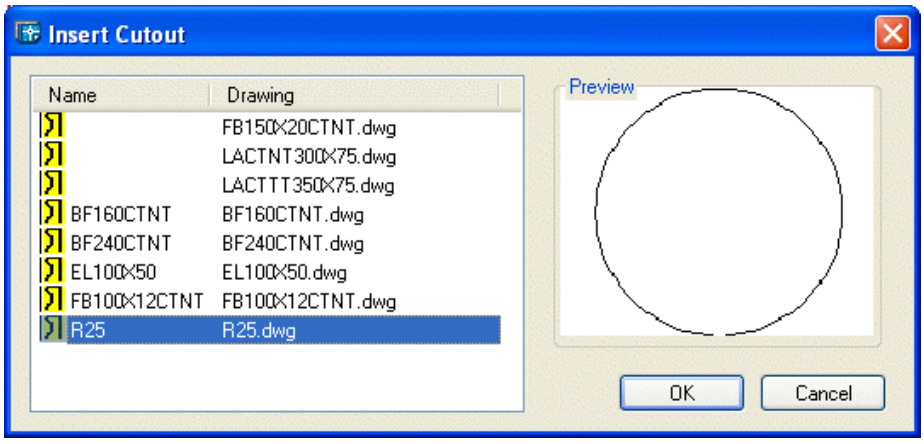
   If the stiffener that you are adding the cutouts to has any mirrored or identical parts, you are asked which of these parts you want to update with the changes you made. See Updating Mirrored and Identical Parts (page 157).

   ShipConstructor updates the cutouts on the stiffener and any selected mirrored and identical parts.

To insert weld reliefs to a stiffener (similar for twisted stiffeners and faceplates)

1. Choose SC Structure > Stiffener > Insert Weld Seam Reliefs (page 419) or right-click on the stiffener and choose Insert Weld Seam Reliefs.
2. Select the stiffener you want to add weld seam reliefs to.
3. Select the start point of the weld seam line. This must be a straight line. If you are using a curved weld seam, pick only one stiffener at a time and use the Osnap Nearest to pick a point close to the point where the weld seam intersects the stiffener. Press Enter.
4. Select the end point of the weld seam line. If using a curved weld seam, then pick only one stiffener at a time and use the Osnap Nearest and pick a point close to the point where the weld seam intersects the stiffener. Press Enter.

   The Insert Cutout window appears.



5. Select the weld insert relief cutout shape you want to use and click OK.

6. If the stiffener that you are adding the cutouts to has any mirrored or identical parts, you are asked which of these parts you would like to be updated with the changes you made. See Updating Mirrored and Identical Parts (page 157).

   ShipConstructor adds the weld seam relief to the stiffener and any selected mirrored and identical parts.

Stiffener Attach to Plate Part

During the creation of a stiffener you can attach it to one or more plate parts but if for some reason a stiffener needs to be attached to a plate part after it has been created this command can be used.

To attach a stiffener to a plate part after the stiffener has been created

1. Choose SC Structure > Stiffener > Attach to Plate (page 414)
2. Select the stiffeners you want to attach to plate parts and Press Enter.
3. Select the plate parts you wish to attach the selected stiffeners to and Press Enter.

   The Select Markline Style window appears.
4. Select the markline style for the stiffener marklines to use on the attached plates and Click OK.

If the plate parts you are attaching the stiffeners to have any mirrored or identical parts, you are asked which of these plate parts you wish to attach stiffeners to as well. For each plate part selected a new stiffener is created and attached to it in the same relative position as the original plate part and stiffener.

ShipConstructor does the following:

- Creates any valid attachments between the selected stiffeners and plate parts.
- Adds a Stiffener marklines to the plate parts where any stiffeners were attached.
- Offsets the position of stiffeners that were attached to the thickness side of any plate parts so they sit on top of the attached plate parts.

Note: Only valid attachments between plate parts and stiffeners will be made. If a stiffener is not in the same plane as the plate part, no connection will be made between this plate part and stiffener.

Stiffener Moldline

To extract the moldline from a stiffener

Choose SC Structure > Stiffener > Extract Mold Line (page 420)

1. Select the stiffener to extract the moldlines from.

   The moldlines of the selected stiffeners are posted to the drawing on the _REV layer.

To extract a neutral axis from a stiffener

Choose SC Structure > Stiffener > Extract Neutral Axis (page 419).

1. Select the stiffener to extract the neutral axis from.

   The neutral axes of the selected stiffener are posted to the drawing on the _Draft_NoProcess layer.

Stiffener Moving

To move a stiffener

A stiffener part can be moved by the following methods:

- Using the AutoCAD **MOVE** command.
- Moving the construction line the stiffener is based on. See Modifying Construction Lines (page 152).
- Moving a plate part the stiffener is attached to. (make sure to move all the plate parts a stiffener is attached to during this operation)

Note: You cannot use the mouse to drag a stiffener part. This operation is disabled within ShipConstructor drawings.

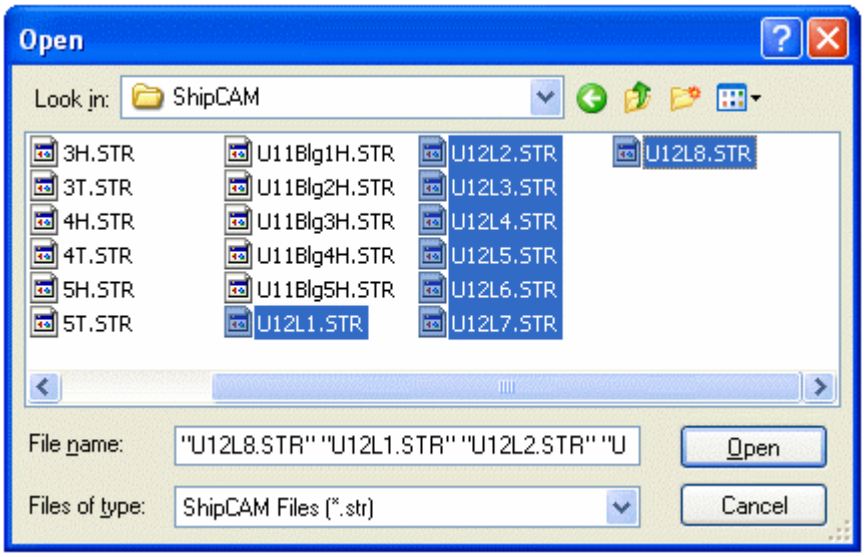
If the stiffener you are moving has any mirrored or identical parts, you are asked which of these parts you want to be moved. Mirrored parts are moved relative to their mirrored axis. See Updating Mirrored and Identical Parts (page 157).

Twisted Stiffeners

A twisted stiffener is a stiffener whose flange orientation twists about its length. These types of stiffeners cannot currently be defined in the same way that regular stiffeners are. ShipConstructor can import ShipCAM .str (stringer) files into a structure drawing or export stringers directly from the hull module as new objects called twisted stiffeners. These twisted stiffeners contribute weight and center of gravity information to the project and can be included in stiffener plots.

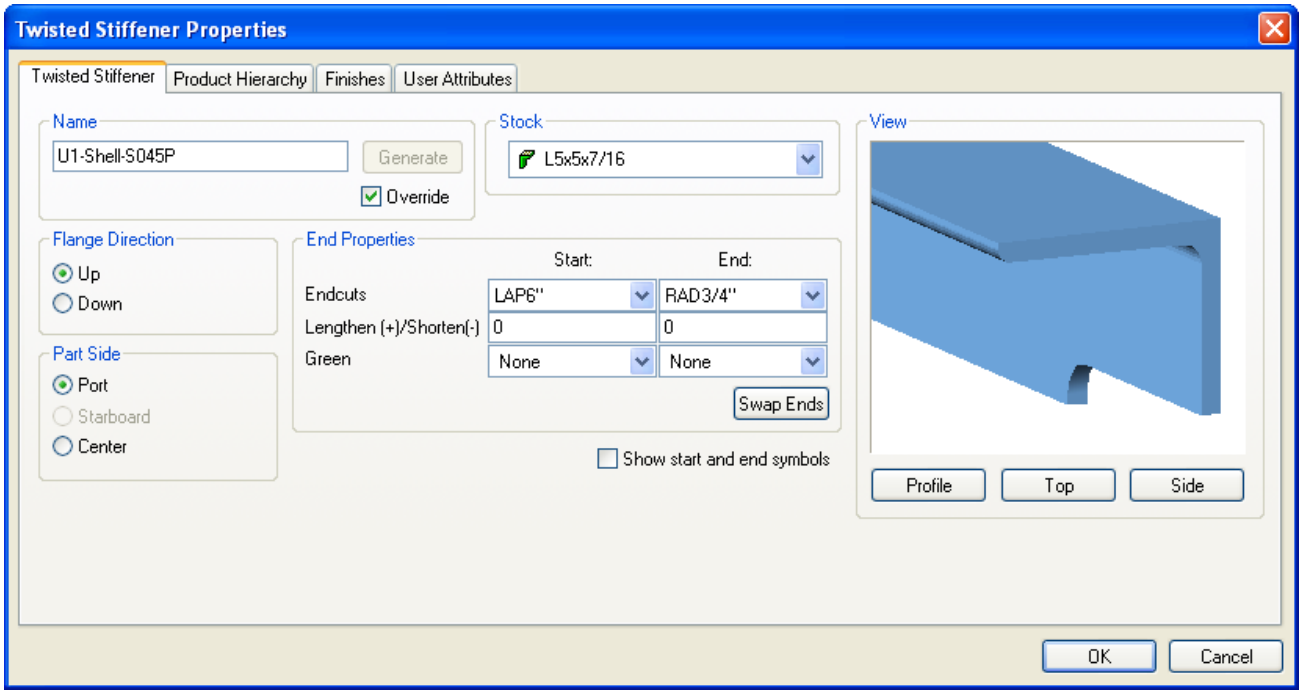
To create a twisted stiffener

1. Choose SC Structure > Stiffener > New Twisted (page 415)
2. Browse to and select the stringer files and then click Open.



Note: The Twisted Stiffener Import Units setting in your Project Settings should match the units that were used when creating the stringer files being opened here. This describes how to change the twisted stiffener import units setting. See Twisted Stiffener Import Units under General > Project Settings (page 327).

The Twisted Stiffener Properties window appears.



2. Set the desired options. See Twisted Stiffener Properties Window (page 415) for details.
3. Click OK.

   ShipConstructor creates the twisted stiffener part.

> Note: ShipConstructor places the stiffener's piecemark on the stiffener in the plane of the web, whereas ShipConstructor2005 places the stiffener piecemark on the plate part.

To export a twisted stiffener from a hull drawing

1. Open a hull drawing with the stringers you want to export as twisted stiffeners.
2. Select SC Hull > Export > To Structure Drawing > Export Stringer or right-click on the stringer and choose Export To Structure.

   The Choose Group Drawing window appears.
3. Select the planar group model drawing that you want to create the twisted stiffeners in and click OK.

   The Twisted Stiffener Properties window appears.
4. Set the desired properties. See Twisted Stiffener Properties Window (page 415) for details.
5. Click OK.

   ShipConstructor creates the twisted stiffener part including the solid in the selected planar group model drawing.

Twisted Stiffener Edit

To edit a twisted stiffener

1. Choose SC Structure > Stiffener > Edit Twisted (page 416) or right-click on the stiffener and choose Edit Properties.

   The Twisted Stiffener Properties window appears.
2. Set the desired properties. See Twisted Stiffener Properties Window (page 415) for details.
3. Click OK.
4. If the twisted stiffener that is being edited has any mirrored parts, you are asked which of these parts you want to update with the changes you made. See Updating Mirrored and Identical Parts (page 157).

ShipConstructor updates the twisted stiffener and any selected mirrored parts using the properties set in the Twisted Stiffener Properties window.

Twisted Stiffener Trimming

To trim a twisted stiffener

ShipConstructor lets you apply more than one trim to each end of the twisted stiffener. The trims can also be applied to specific components of the twisted stiffener (for example, trimming just the flange and leaving the web).

1. Choose SC Structure > Stiffener > Trim (page 418) or right-click on the stiffener and choose Trim.
2. See Step 2 of To trim a stiffener under Stiffener Trimming (page 170)

Twisted Stiffener Cutouts

To add a cutout to a twisted stiffener

ShipConstructor lets you add cutouts to twisted stiffeners. The cutouts can only be applied to the web of the selected part.

See To add a cutout to a stiffener under Stiffener Cutouts (page 171)

To edit a cutout on a twisted stiffener

See To edit a cutout on a stiffener under Stiffener Cutouts (page 171)

Twisted Stiffener Moldline

To extract the moldline from a twisted stiffener

1. Choose SC Structure > Stiffener > Extract Mold Line (page 420)
2. Select the twisted stiffeners to extract the moldlines from.

   The moldlines of the selected twisted stiffeners are posted to the drawing on the current layer.

To extract a neutral axis from a twisted stiffener

1. Choose SC Structure > Stiffener > Extract Neutral Axis (page 419).
2. Select the twisted stiffeners to extract the neutral axis from.

   The neutral axes of the selected twisted stiffeners are posted to the drawing on the current layer.

Twisted Stiffener Moving

To move a twisted stiffener

A stiffener part can be moved by using the AutoCAD **MOVE** command.

Note: You cannot use the mouse to drag a twisted stiffener part. This operation is disabled within ShipConstructor drawings.

If the twisted stiffener you are moving has any mirrored parts, you are asked which of these parts you want to be moved. Mirrored parts are moved relative to their mirrored axis. See Updating Mirrored and Identical Parts (page 157).

Twisted Stiffener Clean Geometry

The purpose of this operation is to allow users to correct issues that can arise from the complexity of the twisted stiffener geometry. This operation may remove points from the parts underlying geometry so users should only use it if they need to. Under normal circumstances there should be no need to use this operation. Possible uses for this operation are:

- To reduce the density of points in the twisted stiffener model to improve performance and reduce complexity.
- To eliminate abnormalities in the geometry which are causing the part to model or plate nest incorrectly.

To clean the geometry of a twisted stiffener

1. Choose SC Structure > Stiffener > Clean Geometry of Twisted (page 416)
2. Select the twisted stiffeners to clean the geometry of.

   The clean procedure is run on the selected twisted stiffeners and any mirrored parts.

Faceplates

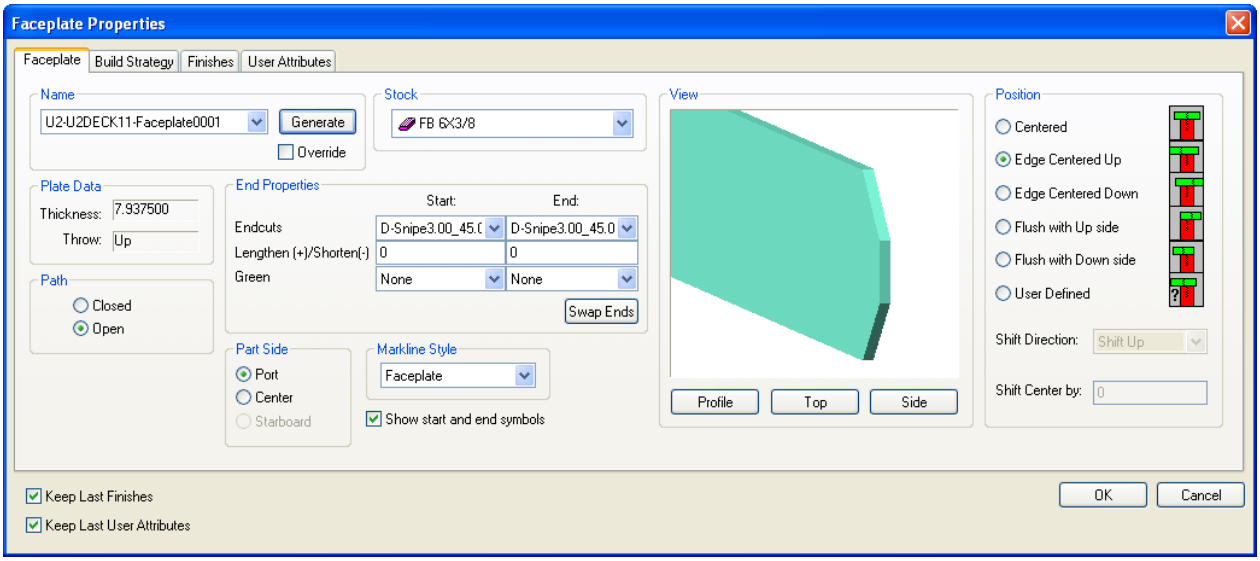
A faceplate is made from flatbar and is attached to the face of a plate part. Faceplates are typically used on the inside of through holes, such as manholes, or on the free side of a plate part to stiffen that edge.

Faceplate Creation

Create a faceplate

1. Choose SC Structure > Faceplate > New (page 420)
2. Select the plate parts to attach the new faceplate to and press Enter.
3. Select the construction lines or AutoCAD curves to create the faceplate on.

   The Faceplate Properties window appears.



4. Set the desired options (see Faceplate Properties Window (page 420) for details).
5. Click OK.
6. Select the start point of the faceplate along the selected path.
7. If Path was set to Closed in the Faceplate Properties window, you are prompted to select another point along the faceplate path to indicate the direction that the faceplate will be created in. The direction is used when calculating the bending information. If Path was set to Open in the Faceplate Properties window, you are prompted to select the end point of the faceplate. If there is more than one segment of the faceplate path connecting the start and end points, you are prompted to select the segment you want to create the faceplate on.
8. Select the thickness direction by picking a thickness point so that the direction from the faceplate start point you selected to the thickness point is in the general direction that the faceplate should throw at the start of the faceplate.

   ShipConstructor does the following:

   - Creates the faceplate part including the solid.
   - Creates a faceplate markline on the plate parts it is attached to.

Faceplate Edit

To edit a faceplate

1. Choose SC Structure > Faceplate > Edit (page 421) or right-click on the faceplate and choose Edit Properties.

   The Faceplate Properties window appears.
2. Set the desired properties. See Faceplate Properties Window (page 420).
3. Click OK.
4. If the faceplate being edited has any mirrored or identical parts, you are asked which of these parts you want to update with the changes you made. See Updating Mirrored and Identical Parts (page 157).

   ShipConstructor updates the faceplate and any selected mirrored and identical parts using the properties set in the Faceplate Properties window.

Faceplate Trimming

To trim a faceplate

ShipConstructor lets you apply more than one trim to each end of the faceplate.

1. Choose SC Structure > Faceplate > Trim (page 422) or right-click on the faceplate and choose Trim.
2. See Step 2 of To trim a stiffener under Stiffener Trimming (page 170)

Faceplate Cutouts

To add a cutout to a faceplate

ShipConstructor lets you add cutouts to faceplates.

1. Choose SC Structure > Faceplate > Cutout > Insert (page 422) or right-click on the faceplate and choose Add Cutout.
2. See To add a cutout to a stiffener under Stiffener Cutouts (page 171)

To edit a cutout on a faceplate

1. Choose SC Structure > Faceplate > Cutout > Edit (page 422) or right-click on the faceplate and choose Edit Cutouts.
2. See To edit a cutout on a stiffener under Stiffener Cutouts (page 171)

Faceplate Moldline

To extract the moldline from a faceplate

1. Choose SC Structure > Faceplate > Extract Mold Line (page 423)
2. Select the faceplate to extract the moldlines from.

The moldlines of the selected faceplates are posted to the drawing on the _REV layer.

To extract a neutral axis from a faceplate

1. Choose SC Structure > Faceplate > Extract Neutral Axis (page 423)
2. Select the faceplate to extract the neutral axis from.

The neutral axes of the selected faceplates are posted to the drawing on the _Draft_NoProcess layer.

Faceplate Moving

To move a faceplate

A faceplate part can be moved by the following methods:

- Moving the construction line the faceplate is based on. See Modifying Construction Lines (page 152).

- Moving a plate part the faceplate is attached to. (make sure to move all the plate parts a faceplate is attached to during this operation)

Note: You cannot use the AutoCAD **MOVE** command or the mouse to drag a faceplate part. These operations are disabled within ShipConstructor drawings.

Corrugated Plates

Corrugated plates are sometimes used to eliminate the need for stiffeners. Corrugated plate parts are created from outer and inner toolpaths and a profile curve.

The profile curve can be defined in two ways:

- In the model drawing, a flat plate stock is used, cut to the expanded size, and then bent to create the corrugated plate.
- From a corrugated plate stock. A standard corrugated plate stock is purchased or bent from flat plate stock prior to cutting.

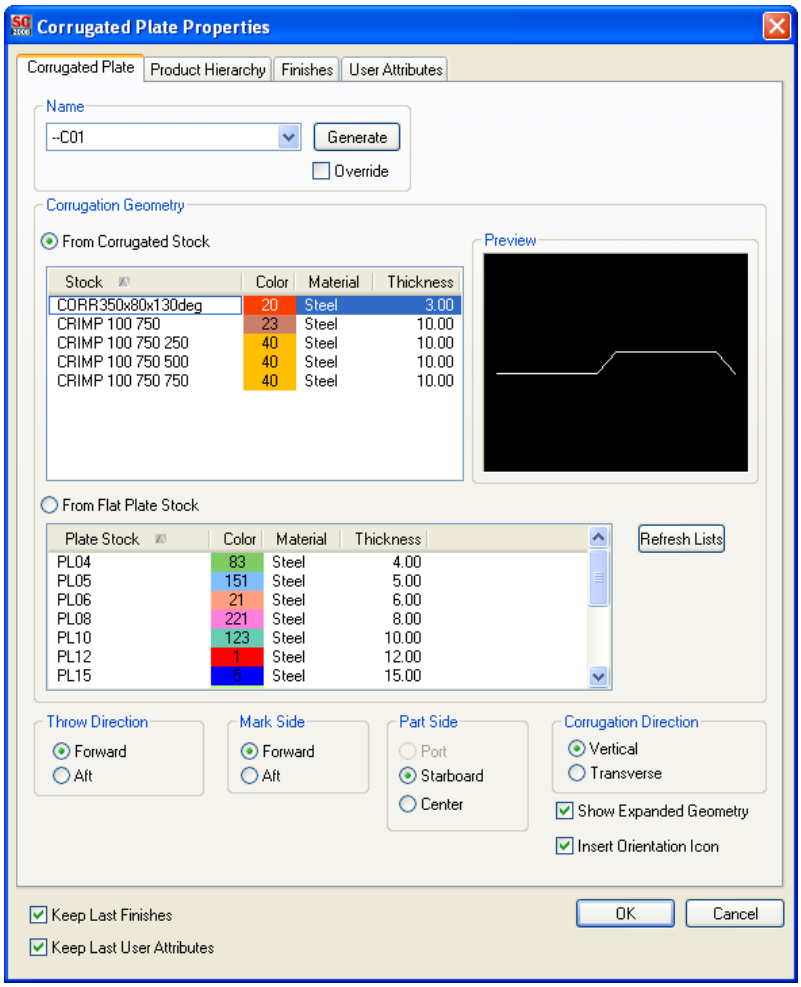
Corrugated Plate Creation

To create a corrugated plate from corrugated stock

Note: Unlike ShipConstructor2005, in ShipConstructor you do not need to define a closed toolpath and extrude a solid before defining a corrugated plate part.

1. Define the boundaries of the part using construction lines or polylines. Use closed polylines to define inner boundaries (holes). The boundaries need to be in the same plane as the current UCS to be considered. The boundary should not be wider than the stock width or longer than the stock length.
2. Choose SC Structure > Corrugated Plate > New (page 423)
3. Select a point within the part boundaries. ShipConstructor automatically detects the part boundaries. This point will be used to place the piecemark as well.

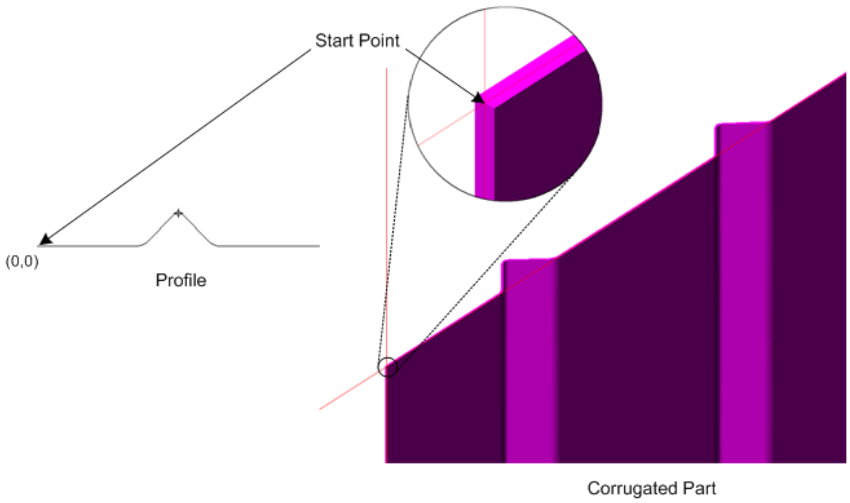
   The Corrugated Plate Properties window appears.



4. Select the From Corrugated Stock option.
5. Set the desired properties. See Corrugated Plate Properties Window (page 424).
6. Click OK.

```
Select Start:
```

7. Select the start point. This point corresponds to the (0, 0) point in the corrugation standard profile. Since the corrugation profile defines the neutral axis line, the solid will be centered on the start point.



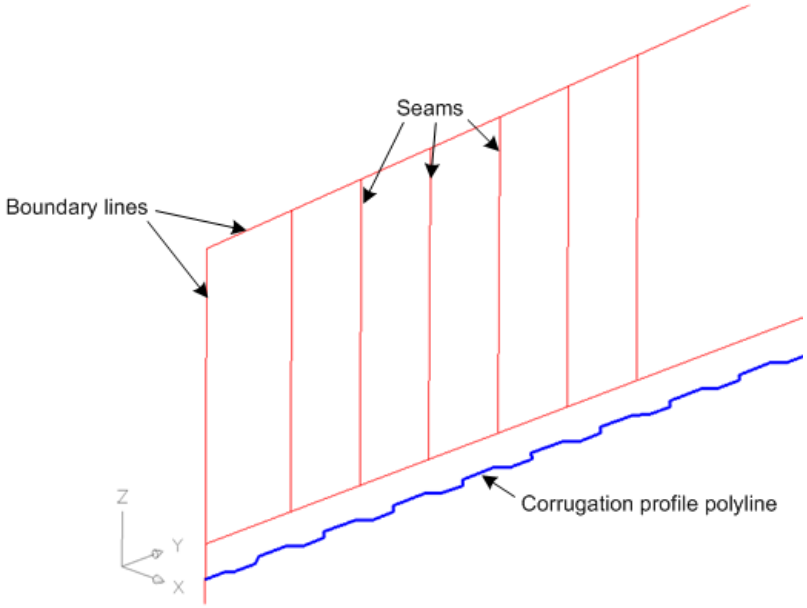
ShipConstructor creates the corrugated plate part.

To create a corrugated plate from flat plate

Note: Unlike ShipConstructor2005, in ShipConstructor you do not need to define a closed toolpath and extrude a solid before defining a corrugated plate part.

1. Define the boundaries of the part using construction lines or polylines. Use closed polylines to define inner boundaries (holes). The boundaries need to be in the same plane as the current UCS to be considered. The boundary needs to include seams.
2. Create or select a UCS that is perpendicular to the boundary plane.
3. Draw the corrugation profile polyline. It is a polyline that indicates the neutral axis of the cross-section of the corrugated plate. The corrugated part will be extruded from the position of the corrugation profile polyline.

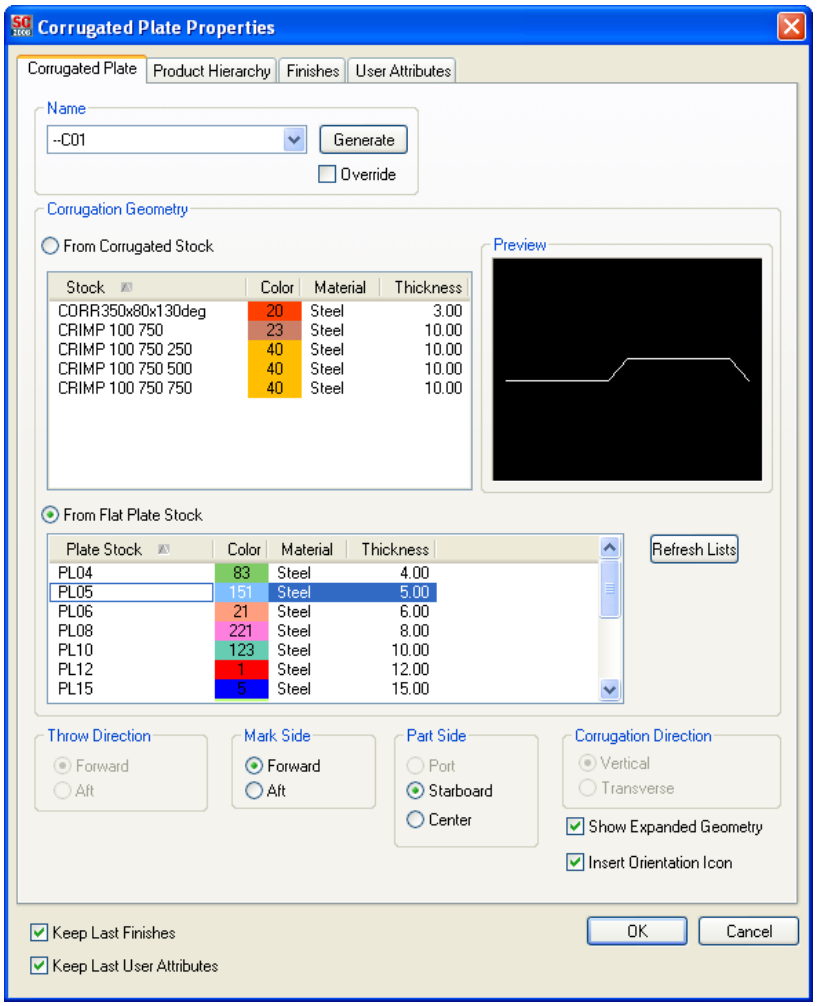
   Hint: If the corrugated bulkhead plate sits on a deck, draw the corrugation profile polyline on the deck.



4. Choose SC Utilities > Activate UCS (page 451) Change the current UCS back to the original plane
5. Choose SC Structure > Corrugated Plate > New (page 423)

6. Select a point within the part boundaries. ShipConstructor automatically detects the part boundaries. This point will be used to place the piecemark as well.

   The Corrugated Plate Properties window appears.



7. Select the From Flat Plate Stock option.
8. Set the desired properties. See Corrugated Plate Properties Window (page 424).
9. Click OK.

```
Select corrugation profile line:
```

10. Select the polyline drawn in step 3.

```
Choose Bend Marker Input Method [Select positions/Auto calculate]<Select positions>:
```

11. Define the fold points along the corrugation profile polyline to be used to generate the foldlines for the expanded production information. This can be done automatically by typing A at the command line and pressing Enter. The foldlines are calculated automatically from the vertex and bend information of the selected corrugation profile polyline. Bend positions will be placed at line segment-to-line segment intersections and at the midpoint of arcs. You can also select points along the corrugation pattern polyline manually to use as the fold points of the corrugated plate part by pressing Enter. If you choose to select positions manually but then press enter without selecting any, the bend positions will be selected automatically.

    ShipConstructor creates the corrugated plate part.

Corrugated Plate Edit

To edit a corrugated plate

1. Choose SC Structure > Corrugated Plate > Edit (page 425) or right-click on the plate and choose Edit Properties.
2. Set the desired properties in the Corrugated Plate Properties window. See Corrugated Plate Properties Window (page 424).
3. Click OK.

   Note: You cannot edit the type of stock a corrugated plate is made of. (For example: changing from a plate stock to a corrugated stock and vise versa.)

4. If the corrugated plate being edited has any mirrored or identical parts, you are asked which of these parts you want to be updated with the changes you made. See Updating Mirrored and Identical Parts (page 157).

   ShipConstructor updates the corrugated plate and any selected mirrored and identical parts using the properties set in the Corrugated Plate Properties window.

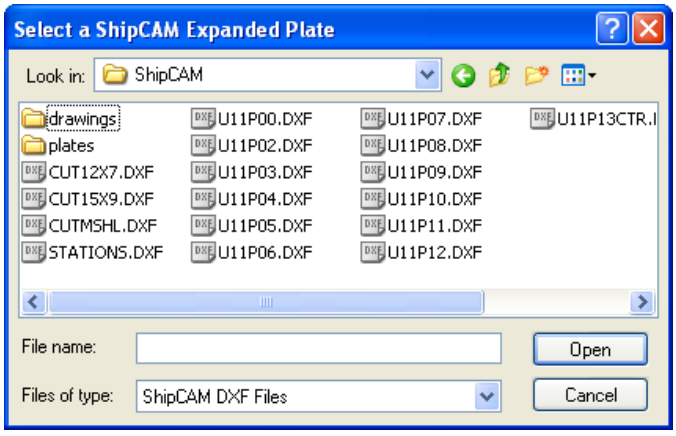
Curved Plates

Curved Plate Creation

Curved plate parts consist of a true 3D representation and an expanded representation for nesting.

To create curved plates from ShipCAM or Maxsurf to a curved model drawing

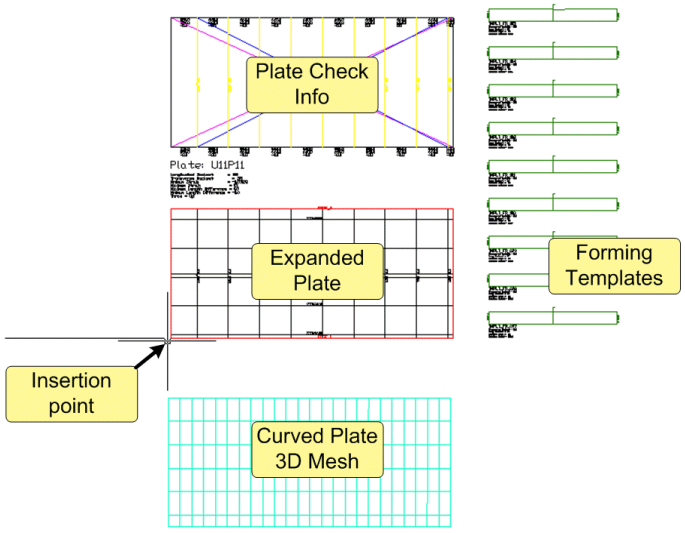
1. Open the curved model drawing that you want to insert the new curved plate parts into.
2. Choose SC Structure > Curved Plate > New (page 426) to open the Select a ShipCAM Expanded Plate window.



   Note: By default, ShipConstructor looks in the Curved folder of the current unit. When exporting expanded plates from ShipCAM, save the exported DXF files to this folder.

3. Select the DXF file containing the expanded and curved plate and then click Open.

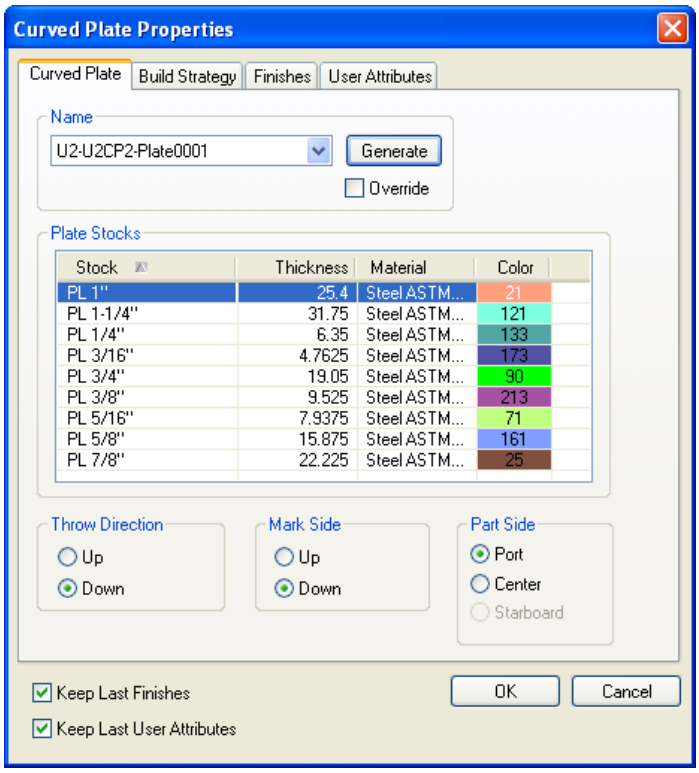
   ShipConstructor inserts objects from the DXF file into the curved model drawing and converts the expanded plate into a curved plate part. Additional objects like plate checking data and forming templates are inserted but not used.



ShipConstructor asks you to specify an insertion point for the expanded plate. By default, ORTHO is on, and the suggested position for the expanded plate and additional information is outboard of the curved plate (see the above image).

4. Move the expanded plate to the desired location.

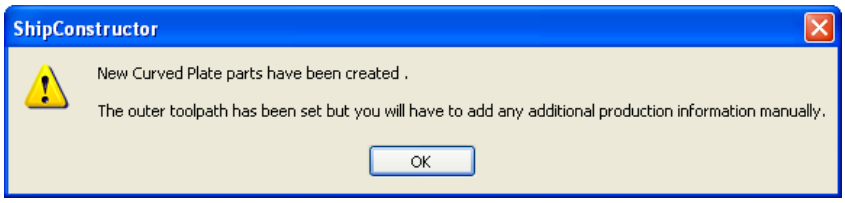
   The Curved Plate Properties window appears.



5. Set the properties. See Curved Plate Properties Window (page 426).
6. Click OK.

   ShipConstructor creates a new curved plate part consisting of the curved surface geometry and the outer toolpath of the expanded plate. Any additional information from the imported DXF file (for example, markings and inner cuts) will need to be added to the curve plate part manually (see To add production information to a curved plate part in

the Curved Plate Production Information (page 189) section below). The name of the part is set to the name of the imported DXF file.


ShipConstructor
New Curved Plate parts have been created .
The outer toolpath has been set but you will have to add any additional production information manually.
OK

To export a curved plate from a hull drawing

1. Open the hull drawing that contains the surfaces you want to export as curved plate parts.

   Note: To export a surface as a curved plate part it must have a stock assigned to it and it must be expanded.

2. Choose SC Hull > Export > Export To Structure.

   The Choose Group Drawing window appears.

3. Select the curved plate drawing that you want to create the curved plate parts in and click OK.

   The Curved Plate Properties window appears.

   Note: If more than one surface was selected to be exported the Curved Plate Properties window will appear once for each surface. The view will be zoomed to the current surface and the surface will be highlighted in the hull drawing.

4. Set the desired properties. See Curved Plate Properties Window (page 426).
5. Click OK.

   ShipConstructor creates a new curved plate part consisting of the curved surface and expanded geometry. All toolpaths and markings from the selected surfaces are exported to the new curved plate parts using the styles and colors defined in Manager. Once a hull surface has been exported as a curved plate part there is no link kept between the surface and the curved plate part. Any changes made to the surface after the curved plate part had been created will not be reflected by the curved plate part. If stock was added during the surface expansion the NURBS curve defining the expanded boundary with the added stock is used as the outer toolpath of the curved plate part. The expanded boundary without the added stock is added as a marking to the curved plate part. All forming templates for the exported surfaces are copied to the destination curved plate drawing and are placed on the _Draft_Cut layer so they can easily be made into plate parts if needed. Plate check text for the exported surfaces data is copied to the destinations drawing as AcDbMText and placed on the _Draft_NoProcess layer.

   Note: If the expanded hull surface contains an orientation icon, its axes will be added to the curved plate part as a markline using the Orientation Icon markline style. The axes' labels are added as markline text.

   Note: Only visible markline labels on the hull surface will be added to the curved plate part as markline text.

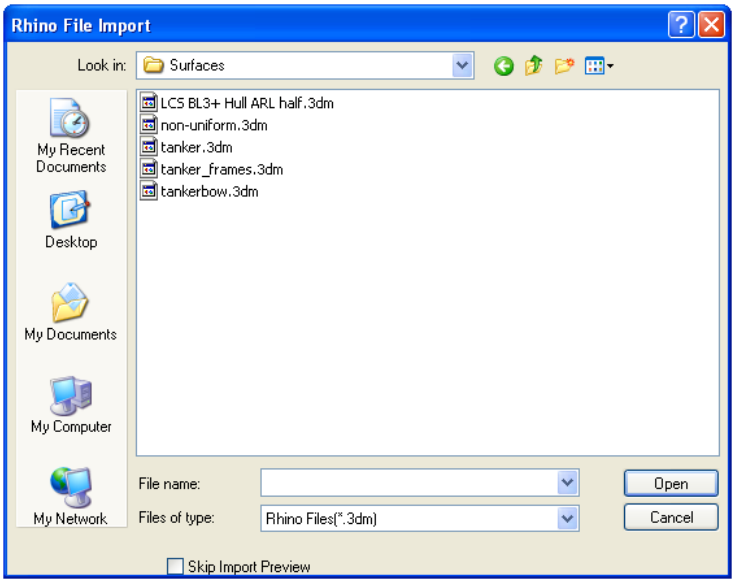
   The following markline styles exist for use by Hull surfaces and curved plate part marklines:

   - Projection Line
   - Section Line
   - Girth Line
   - Expansion Line
   - Roll Line
   - Curved Plate Green
   - Iso Curvature Line
   - Iso Strain Line
   - Forming Control
   - Template Alignment Icon
   - Orientation Icon

To create a curved plate from a Rhino file

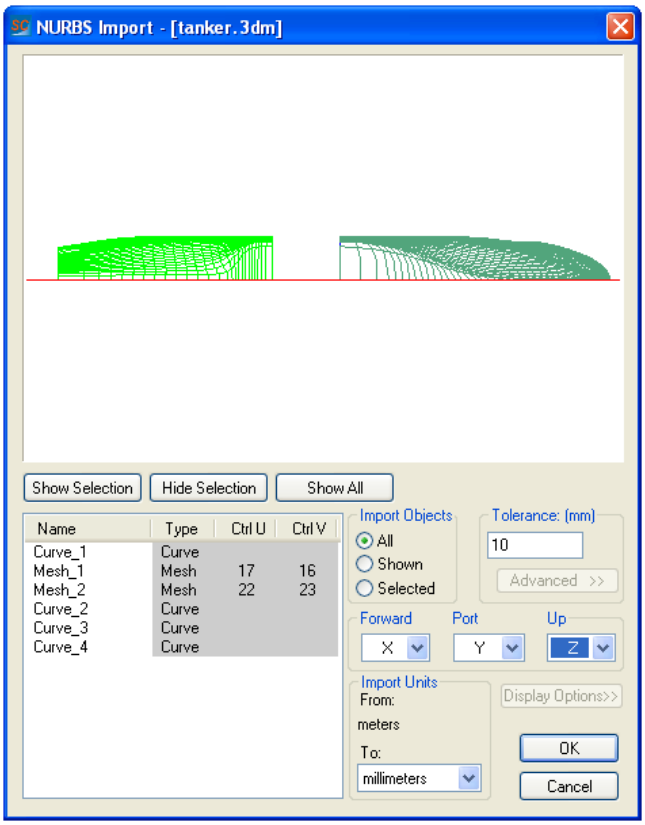
Note: This will create a curved plate solid with invalid production geometry. The intention is that Rhino surfaces can be imported for visualization purposes only, without requiring a license for the Hull module.

1. Open the curved model drawing that you want to insert the new curved plate parts into.
2. Choose SC Structure > Curved Plate > Import Curved Plate From Rhino (page 428) to open the Rhino File Import window.



3. Select the 3dm file containing the desired curved surfaces and then click Open.
4. The NURBS Import window will appear. Set the desired properties in the NURBS Import window, then click OK.

Note: The curves listed in this window will not be imported into the drawing. Only the meshes will be imported, and they will be converted into curved plates automatically.



5. The Curved Plate Properties window will appear. Set the desired properties and click OK. This window will appear once for each imported curved plate.

   See Curved Plate Properties Window (page 426).

   ShipConstructor creates a curved plate part consisting of the curved surface geometry and an X-shaped invalid outer toolpath, representing the expanded plate.

Curved Plate Edit

To edit a curved plate

1. Choose SC Structure > Curved Plate > Edit (page 427) or right-click on the plate and choose Edit Properties.

   The Curved Plate Properties window appears.

2. Set the desired properties. See Curved Plate Properties Window (page 426).
3. Click OK.
4. If the curved plate being edited has any mirrored parts, you are asked which of these parts you want to update with the changes you made. See Updating Mirrored and Identical Parts (page 157).

   ShipConstructor updates the curved plate part and any selected mirrored parts using the properties set in the Curved Plate Properties window.

To define a new expanded outer toolpath for a curved plate part

1. Choose SC Structure > Curved Plate > Define New Outer Toolpath (page 428)
2. Select the curved plate part to define a new expanded outer toolpath for.

   Note: The new expanded outer toolpath must be a closed AutoCAD curve or construction line.

3. If the curved plate being edited has any mirrored parts, you are asked which of these parts you want to update with the changes you made. See Updating Mirrored and Identical Parts (page 157).

ShipConstructor replaces the old expanded outer toolpath with the selected new one in the curved plate and any selected mirrored parts.

Note: The curved surface is not updated to reflect the new geometry of the new expanded outer toolpath.

Curved Plate Production Information

To add production information to a curved plate part

1. Choose SC Structure > Curved Plate > Add Objects
2. Select the production entities you want to add to the curved plate part and press Enter. Production entities are added as follows:
   - Closed AutoCAD curves or construction lines on the _Draft_Cut layer are added as inner toolpaths.
   - AutoCAD curves or construction lines on the _Draft_Mark or _Draft_NoProccess layer are added as markings.
   - AutoCAD text entities are added as mark text.
   - SConCurvedPlateMarklines entities are added as markings.
   - SConCurvedPlateMarktext entities are added as mark text.
3. Select the curved plate part to add the production entities to.
4. If the curved plate being edited has any mirrored parts, you are asked which of these parts you want to update with the changes you made. See Updating Mirrored and Identical Parts (page 157).

   ShipConstructor adds the selected production information to the curved plate and any selected mirrored parts.

To extract production information from curved plate parts

1. Choose SC Structure > Curved Plate > Extract Production Info (page 428).
2. Select the curved plate parts to extract the production information from.

   ShipConstructor copies the production entities from the selected parts and posts them to the current drawing. The production entities include the outer toolpath, inner toolpaths, marklines and mark text.

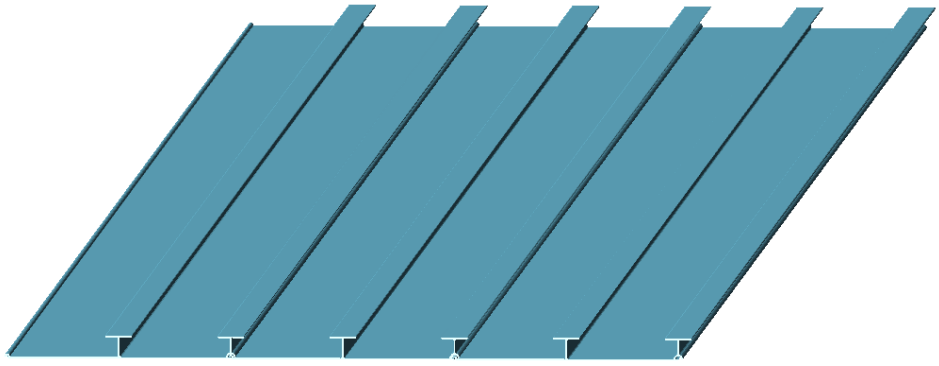
To move the production information of curved plate parts

1. Select the curved plate part whose production information you want to move.
2. Click one of the grip points of the production information of the selected curved plate part.
3. Select the desired new location for the curved plate's production information.

   ShipConstructor moves the production information portion of the curved plate part to the selected new position and leaves the curved surface portion of the curved plate part where it is.

Planks

Planking (or automatic straking) automatically divides predefined areas, such as decks and bulkheads, into planks based on the parameters of available stock and then generates a bill of materials (BOM) automatically. Previously, straking had to be done manually, so an automatic method of straking represents a significant saving of time. Automatic straking also works with prefabricated aluminum extrusions and sandwich panels.



Extruded panels

Planks Creation

To create planks

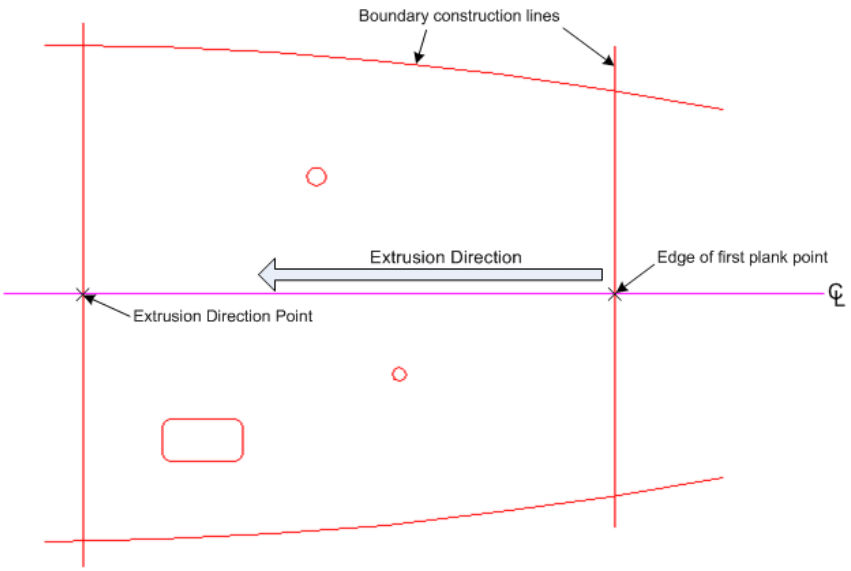
> Note: Unlike ShipConstructor2005, you do not need to define a closed toolpath and extrude a solid before defining plank parts.

1. Define the boundaries of the part using construction lines, or polylines. Use closed polylines to define inner boundaries (holes).
2. Choose SC Structure > Planks > New (page 428)

   ```
   Select interior point:
   ```
3. Click within the part boundaries. ShipConstructor automatically detects the part boundaries. This point is also used to place the piecemark.

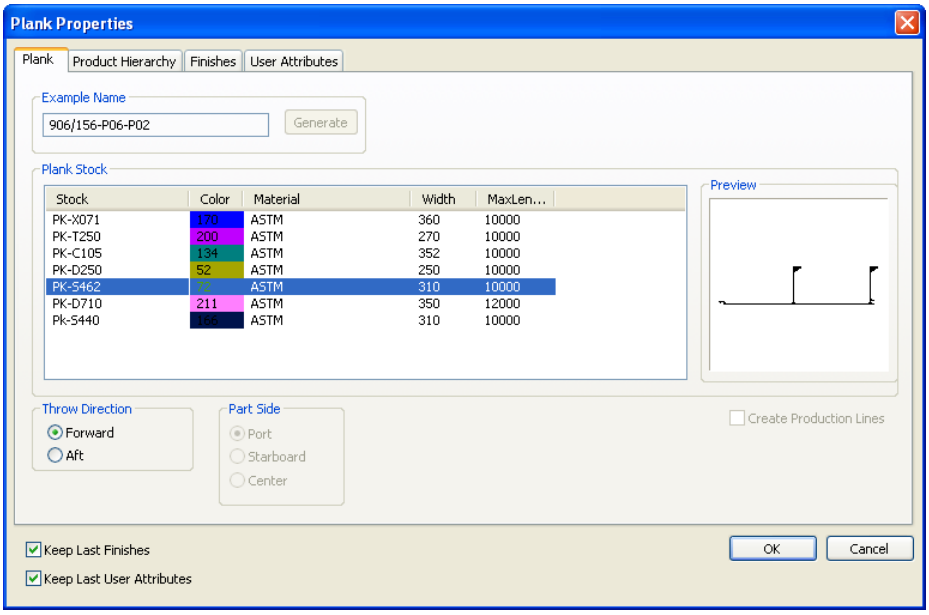
   ```
   Select a point on the outer toolpath indicating the edge of the first plank:
   ```
4. Select a point on the outer trim loop. This point will be the starting point for inserting planking. It will be aligned with the (0, 0) point in the planking profile drawing.

   ```
   Select extrusion direction point:
   ```
5. Select a second point. The line formed between the first and the second point indicates the direction along which the planks will be extruded.



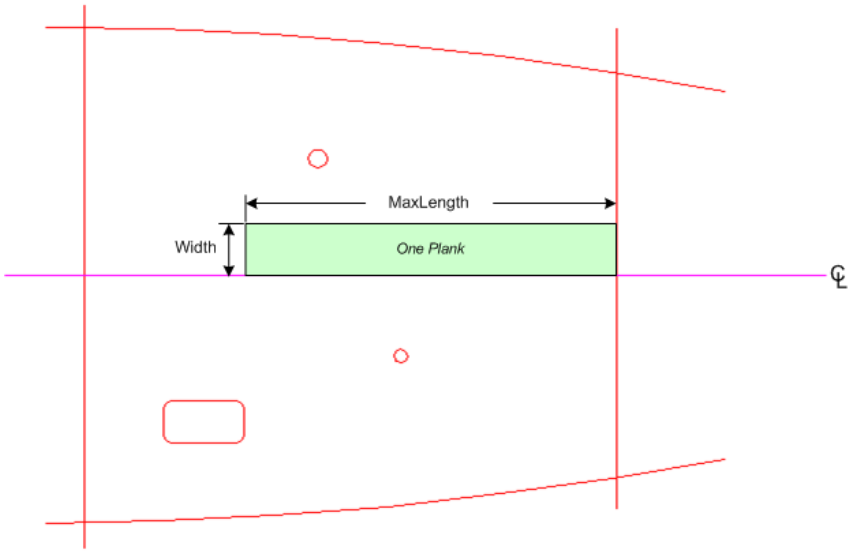
Typical planked deck

The Plank Properties window appears.



6. Set the desired properties. See Plank Properties Window (page 429).
7. Click OK.

   ShipConstructor fills in the part boundaries with plank parts.



Planks are inserted automatically starting from the insertion point. If the outer toolpath is larger than the maximum length of the planks, additional planks are inserted.

Planks Edit

You change the properties of a plank collection after it has been created.

To edit planks

1. Choose SC Structure > Planks > Edit Collection (page 430) or right-click on the plank and choose Edit Collection Properties.

2. Select a plank in the planked area that you want to edit.

   Note: You cannot edit the properties of individual planks. Any changes you make will be made to all the planks in a planked area.

   The Plank Properties window appears.

3. Set the desired properties. See Plank Properties Window (page 429).

   Note: You cannot edit the part side of the plank collection.

4. Click OK.

5. If the planking being edited has any mirrored parts, you are asked which of these parts you want to update with the changes you made. See Updating Mirrored and Identical Parts (page 157).

   ShipConstructor refills the part boundaries with plank parts. Most changes are performed on the existing collection however some changes, like updates to the plank stock, require that the entire plank collection be rebuilt. A rebuild is necessary when changing to a plank stock with a different width or a shorter maximum length than the existing plank stock. If the maximum length of the new plank stock is greater than that of the existing stock, an additional option is presented to allow you to reuse the existing layout. Rebuilding a plank collection will automatically regenerate any planks that were manually deleted from the collection.

   Note: Any changes that have been made to the position of the ends of the individual plank parts will be lost when the plank parts are regenerated.

To modify planks using their grip points

ShipConstructor planking solids are intelligent objects that are logically connected to each other. If one plank is moved using the grip points, adjacent planks will stretch and compress. This allows for individual planks to be lined up to existing structural parts and the staggering of planks within a deck.

1. Select the plank part you want to modify.

2. Select the grip point at the connection between two plank parts and move it to the desired location.

   Note: Shift-click to select multiple grip points at the same time

   While modifying grip points, planks are stretched and compressed as needed. If there is not enough planking stock, new planks are inserted as needed.

Adding and Removing Cut Lines to a Plank Collection

You can add and remove cut lines to an existing plank collection by using the Add Objects and Remove Objects commands. More than one plank collection can be selected for this command and more than one cut line can be added to each collection. Cut lines that do not intersect any of the selected plank collections are ignored.

To add a cut line to a plank collection

1. Choose SC Structure > Plate > Add Objects (page 404)

2. Select all plank collections that you want to add cuts to.

3. Select the objects to add to all selected plank collections.

   The selected cuts are added to the existing collection and the affected plank parts are updated. Previously identical plank parts within a collection may be renamed if their outlines change.

   Note: Plank parts that are completely obscured by cut lines are removed from the plank collection and will not be restored if the cut line is removed.

To remove an object from a plank collection

1. Choose SC Structure > Plate > Remove Objects (page 406)

2. Select the cut line on the plank collection that you want to remove.

   The selected cut is removed from the existing collection and the affected plank parts are updated. Previously dissimilar plank parts within a collection may be renamed if their plank outlines change.

Splitting Plank Collections

Splitting is a way to define a new plank collection from a subset of planks in an existing collection. The remaining plank collection is reduced by the set of planks used to make the new collection. Note that both the new plank collection and remaining plank collection retain the original area of the plank collection prior to the split.

To split a plank collection

1. Choose SC Structure > Planks > Split Plank Collection (page 430)
2. Select one or more planks within a plank collection.
3. Press Enter to complete the operation.

> Note: Splitting a plank collection will break any previously existing relationships between planks in the old and new collections. Plank names in the new collections may change as a result of this operation.

Deleting Plank Collections

To delete plank collections

1. Choose SC Structure > Planks > Delete Plank Collection (page 430)
2. Select a plank in each of the planked areas that you want to delete. A confirmation window appears.
3. Click Yes to continue or No to cancel the operation and return back to the drawing.

Standard Parts

Defining standard parts is an easy way to create typical parts without having to create the geometry for them every time you want to use the part. These will typically be brackets, chocks, clips, and collars.

To insert a standard part

1. Make sure that you are in the UCS you want the part located in.
2. Choose SC Structure > Insert Standard Part (page 444)

```
Specify insertion point or [XMirror/YMirror/Change throw]:
```

3. Select the insertion point. The part is inserted with the origin in the standard drawing at the picked position.

```
Specify direction point or [XMirror/YMirror/Change throw/SNap<on>]:
```

4. Use the command line options to manipulate the part to the desired orientation.
5. Rotate the part to its final position using the cursor.

Standard Brackets Attached to Stiffeners

To prepare for inserting the standard parts you will need to show the parts you want to attach the bracket to in the current drawing. MLink the drawings that contain parts you want to select that are not in the current model drawing. The command can work in any structure drawing. You will need to be in the drawing where you want the standard parts to be saved.

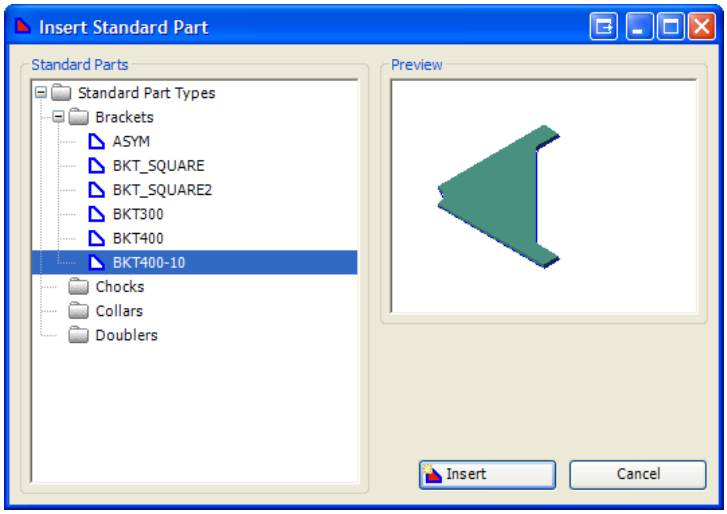
1. Choose SC Structure > Insert Standard Part at Stiffener (page 445)

```
Select primary stiffener:
```

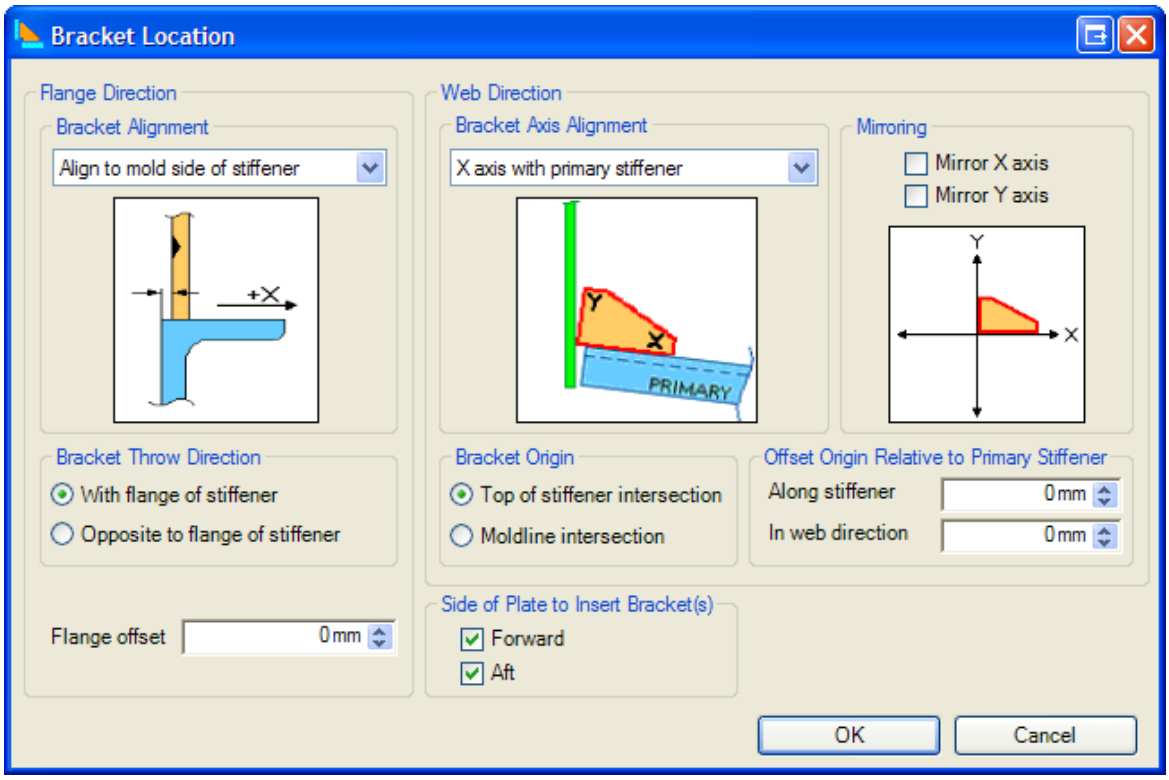
2. Select primary stiffener (stiffener or twisted stiffener)

```
Select secondary part [ENTER for none]:
```

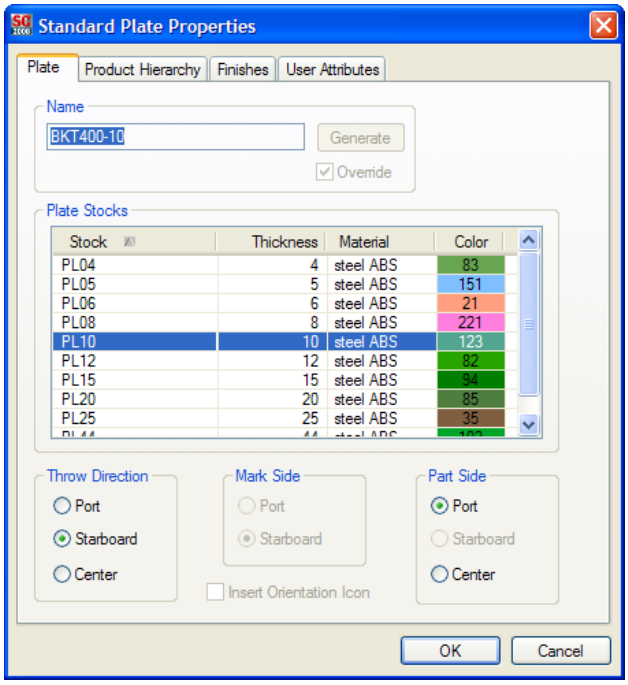
3. Select secondary part (stiffener, twisted stiffener, or plate). Pressing ENTER for none will place the bracket at the end of the stiffener nearest to the pick point of the primary stiffener.
4. Select the Standard Part to insert



5. Select the parameters to determine where to insert the part



6. Show Part Edit Dialog or the first part only. Subsequent parts will be assigned the same product hierarchy and have the same attributes.



7. Throw direction does not need to be changed.
8. Repeat the selection of parts for inserting more brackets. With the same options as the first selection.

Mirror Parts

Mirror Parts About the Centerline

You can mirror one or more parts about the centerline. Each part automatically receives a new part name. You can swap text strings in the mirrored parts, as indicated in the window below.

To mirror a part about the centerline

1. Choose SC Utilities > Mirror about Centerline (page 461)
2. Select all objects that you want to mirror and then press Enter.

   The parts are mirrored to the desired side. All mirrored parts are assigned new names automatically. The project database is also updated with the new parts. Using the above window as an example, the word PORT will be replaced with STBD in the starboard versions of the parts.

Mirror Catamaran Hulls

You can mirror parts for catamaran hulls saving hours of work when modeling twin-hulled vessels. The AutoCAD **MIRROR** command lets you mirror about any line.

Piecemarks

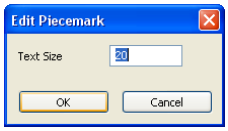
Changing a Piecemark Size

You can change the size of an individual part's piecemark or change the size of all piecemarks of a type.

Note: Piecemark sizes that have been overridden are not affected by changing the piecemark standard size.

To change a piecemark size on one part

1. Choose SC Structure > Parts > Edit Component (page 435)
2. Select the piecemark. If the selected part has identical or mirrored parts then you will see the Update Related Objects window. Select the parts you want to change.
3. The Edit Piecemark window appears.



4. Change the Text Size and click OK. The piecemark size is updated for that part. Now, changing the standard piecemark size will not affect this part.

To change all piecemark sizes

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose Structure > Piecemark Styles (page 345) to open the Piecemark Styles window.
3. Double-click the Text Size field and enter a new value.
4. Click OK to close the Piecemark Styles window.
5. To see the updated piecemark size in open drawings, run SC Utilities > Reload Drawing (page 462)

Change a Piecemark Position

You may find it necessary to move a piecemark to avoid overlapping marks or to make the part information easier to read.

To move a piecemark position

1. Select the part. The part's grip points appear.

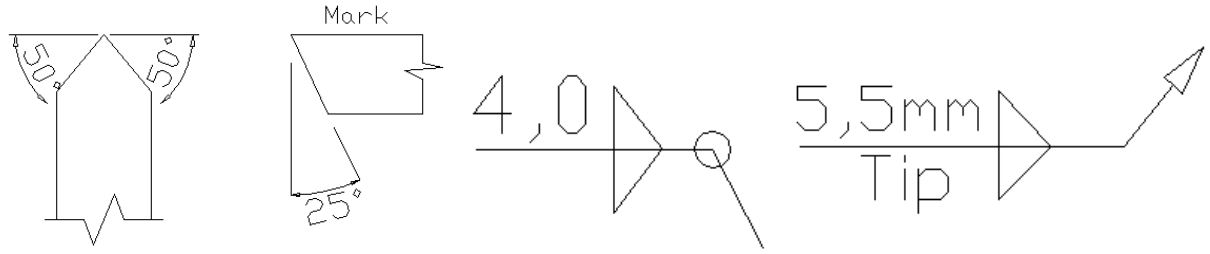
- X -- Move grip point
- -- Rotate grip point

2. Select the move grip point. It will turn to the selected grip color (red).
3. Move the part to the desired location. Do not place the piecemark outside the boundary of the part. This will cause the part to be invalid (see Invalid Parts (page 209)). If you do move the piecemark outside the part or if the part's boundary changes so that the piecemark is outside the part, move the piecemark inside the part.

Note: The piecemark point is used as the location of the part in the nest drawing. If a part is already nested, you should open the nest drawing and re-nest all parts that have changed. Make sure to check all parts with changed piecemarks, as the part position will have shifted and it might interfere with parts next to it.

Dynamic Marking Blocks

A dynamic marking block (DMB) is a grouping of polylines and text with dynamic elements that can be placed and positioned on a structure part as if it was a single entity. This makes it much easier to edit, update, and maintain the markings on multiple parts. The following are several examples of DMBs:



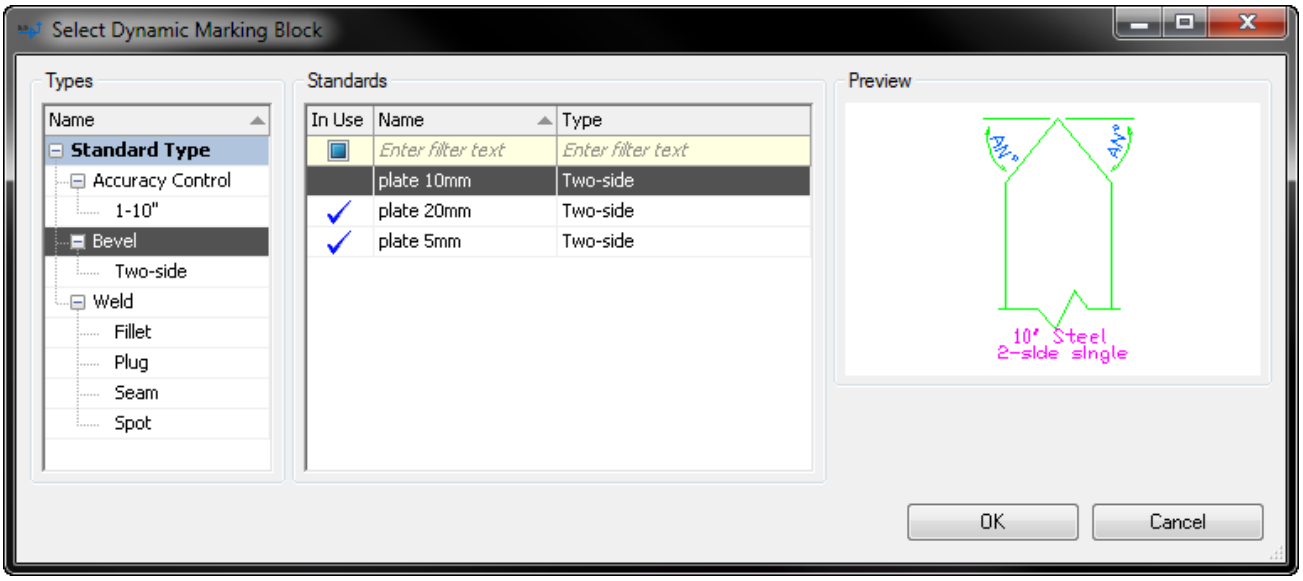
Dynamic marking blocks are created through the dynamic marking block catalog in manager (see Dynamic Marking Block Standards (page 101)).

Add a Dynamic Marking Block

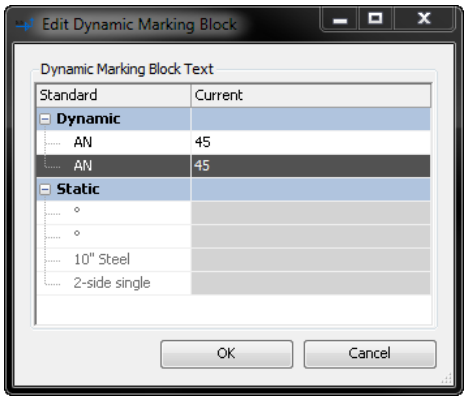
After at least one dynamic marking block standard has been created, a DMB can be added to a structure plate, corrugated plate, or curved plate. Currently these are the only parts that support dynamic marking blocks.

To add a DMB to a part:

1. From the structure menu choose SC Structure > Parts > Add Dynamic Marking Block (page 438)
2. Select the plate, corrugated plate, or curved plate. The Select Dynamic Marking Block dialog is displayed.



3. Select the desired DMB. The standard list can be filtered down by selecting a standard type from the Types table.
4. Click OK. The DMB will be displayed with the origin of the DMB standard positioned at the cursor.
5. Use the command line options to <Mirror/Rotate/Scale> this instance of the DMB.
6. Position the DMB with the mouse to the correct location on the part and click to insert. If there is dynamic text in this DMB, the Edit Dynamic Marking Block dialog is displayed.



7. Edit the dynamic text if desired and click OK. The dynamic marking block will be added to the part.

Edit a Dynamic Marking Block

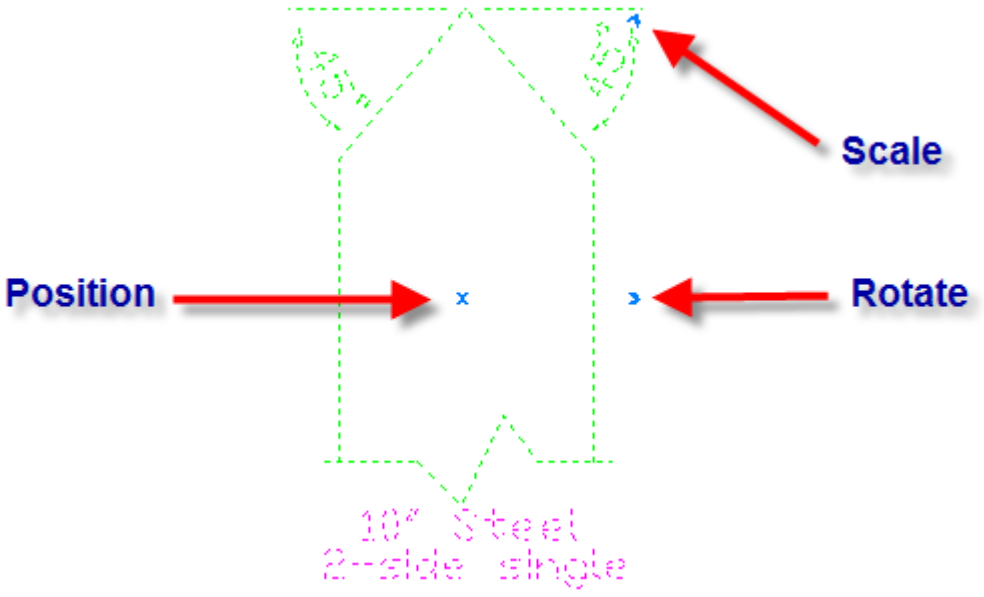
Several properties of a dynamic marking block can be edited anytime after insertion.

TEXT EDIT

Dynamic text on the DMB can be edited any time with the edit component functionality. From the structure menu, select SC Structure > Parts > Edit Component (page 435) and click on a dynamic marking block to bring up the Edit Dynamic Marking Block dialog (shown above).

GRIP POINT EDIT

Each dynamic marking block contains grip points that can be used to manipulate the DMB instance. Grip points are displayed when the part containing the dynamic marking block is selected. The function of each grip point is shown below:



Remove a Dynamic Marking Block

Dynamic marking blocks are removed from parts with the standard remove objects functionality. Select SC Structure > Plate > Remove Objects (page 406) from the parts menu then select the DMB to remove.

Part User-Defined Attributes

User-defined attributes are values that can be associated with any part type. The list is defined in Manager. Those attributes can be assigned to each part type. And then that attribute can be either be required or not required for the part type. The required state has three levels to it: required and deferrable, required and not deferrable, and required default deferred. Deferrable attribute can be deferred for later; therefore, it's not required to provide a value during the creation (or editing). However, a value will need to be provided later.

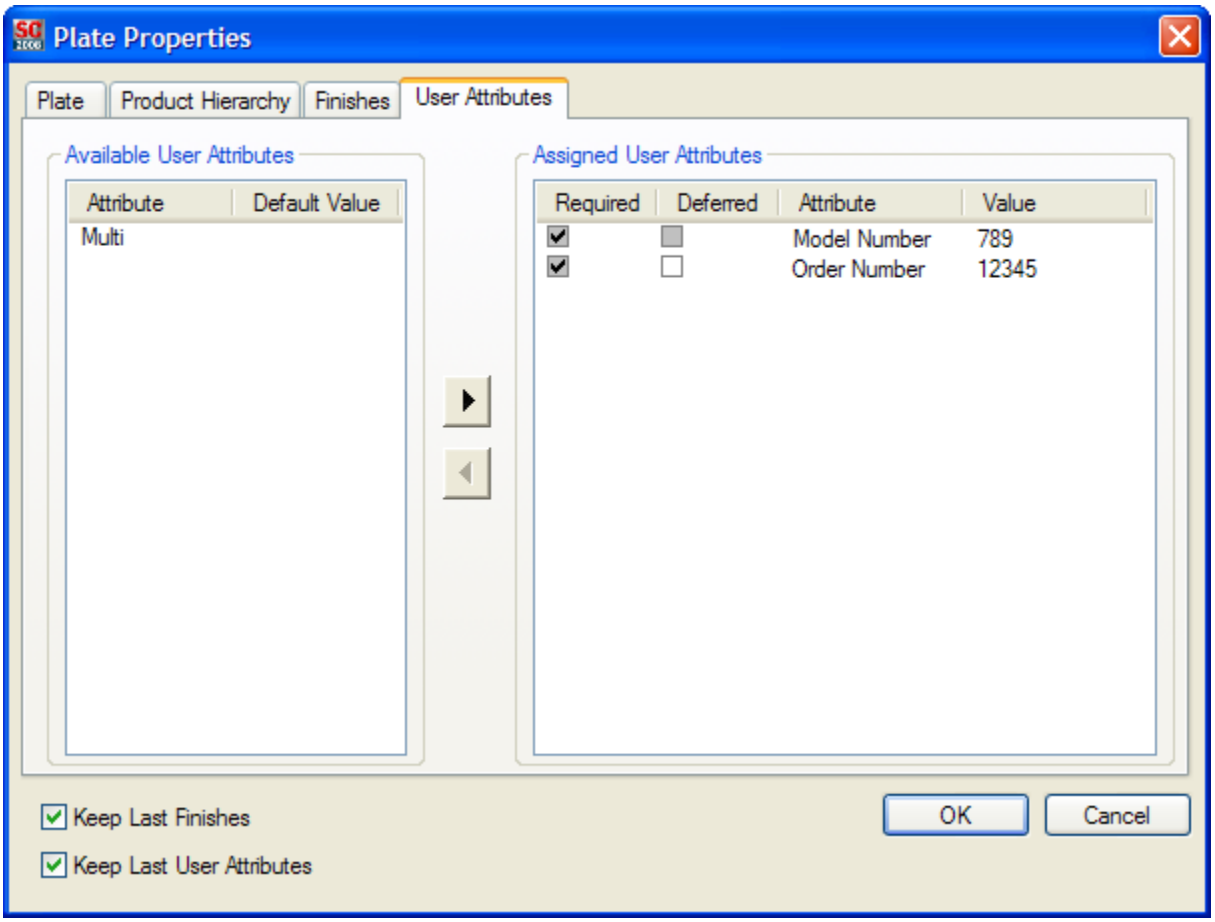
Required attributes are automatically assigned to a part during creation. The deferred state is set if the attribute is required default deferred. The default value will also be set, if one was provided. Attributes set as required and not deferrable must have a value before the properties window can be closed. This also applies to any required and deferrable or required default deferred attributes without the "deferred" state set and no value.

Providing a value allows the user to continue, or simply set the "deferred" state (for those attributes that can be deferred). The "deferred" state checkbox is gray for required and not deferrable attributes.

The left side of the properties window lists all the available attributes. Only non-required attributes can be in the list. Default values will be used, if set. Providing/editing the value for a deferred attribute will clear the deferred state.

The right side lists all the attributes assigned to this part, along with their required and deferred states, and attribute value. Only non-required attributes can be removed from this list.

The following is an image of the User Attributes page in the Part Properties window:



All related parts will have the same user attributes. Adding or removing an attribute to a part will also add or remove that attribute to all related parts. However, the values do not have to be same between related parts.

Check Interferences

Interference checking is a vital step in ShipConstructor that is used to detect part collision problems or to add penetrations for pipe and HVAC through structure.

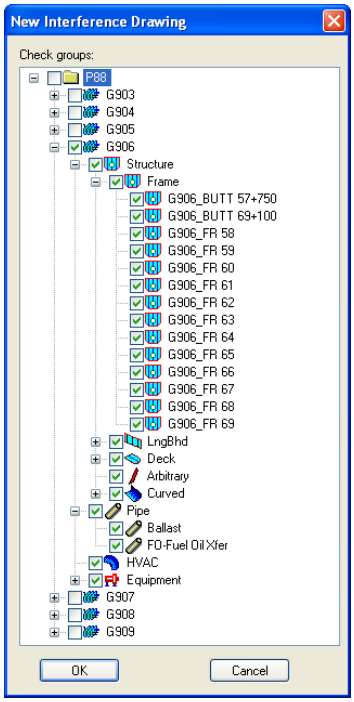
The interference drawing can also be used as a virtual reality fly-through.

Create an Interference Drawing

To create an interference drawing

1. In Navigator, select the Interference page.
2. Click New.

   The New Interference Drawing window appears.



3. Click the check box beside the drawings you want to include in the interference drawing.
4. Click OK.

Checking Interferences

Typically, you create an interference drawing that includes several planar group drawings within it and then check interferences within the interference drawing. (You can also check interferences from within a planar group drawing.) ShipConstructor then checks the solids within the current drawing for collisions. (ShipConstructor does not check Xref solids.)

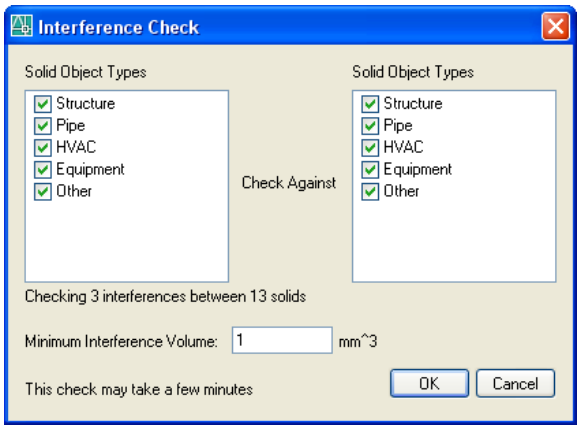
Running the List command on the interference solid will list the interfering part names, and the volume and extents of the interference.

To check interferences

1. Create an interference drawing that includes the planar group drawings you want to check (see Create an Interference Drawing (page 200)).
2. Open the interference drawing you want and select an isometric viewpoint.
3. Choose SC Interference > Check Interferences (page 510) to open the Interference List window.

   Note: You can continue using ShipConstructor with the Interference List window open. For example, you may want to change your viewpoint without closing the Interference List window. You can also close the Interference List window and re-open it by choosing SC Interference > Check Interferences (page 510).

4. Click Run Check.



   The two Solid Object Types lists let you control which types of objects to check for interferences.

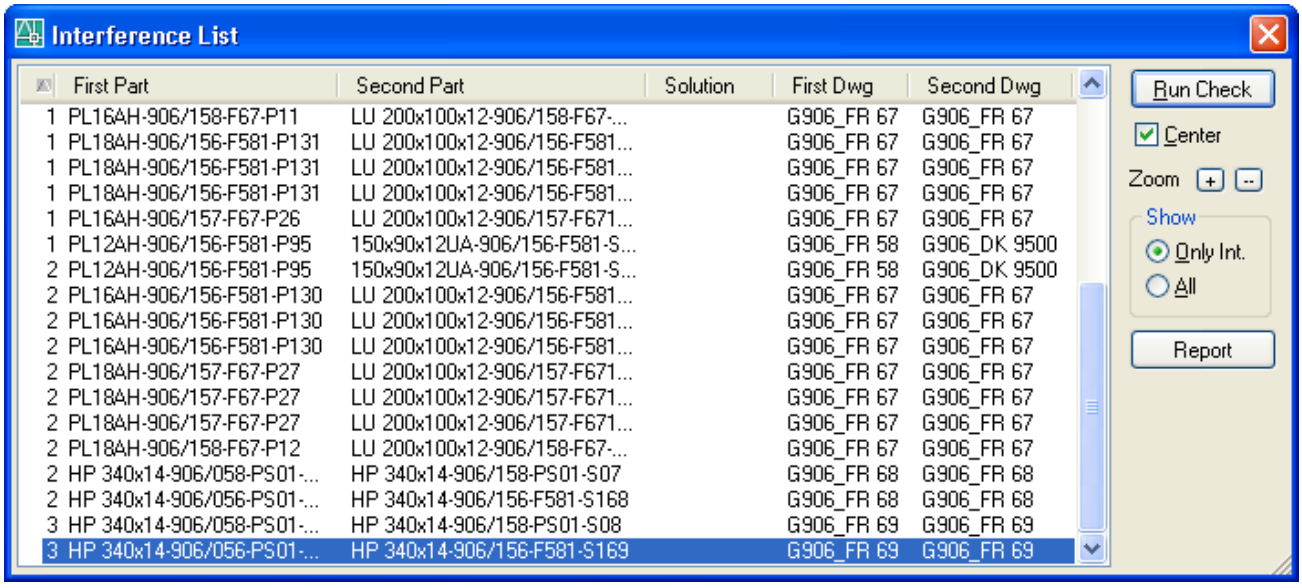
   Below the two lists is the number of possible interferences that ShipConstructor will check.

   The Minimum Interference Volume lets you ignore very small interferences that might be due to computer rounding errors or are so small that they would not be noticed during construction.

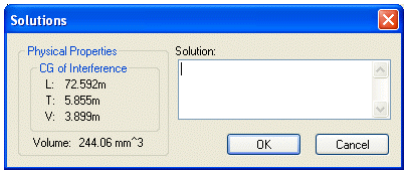
5. Click OK to start the interference check.

   The check may take several minutes.

   ShipConstructor lists interferences in the Interference List window.

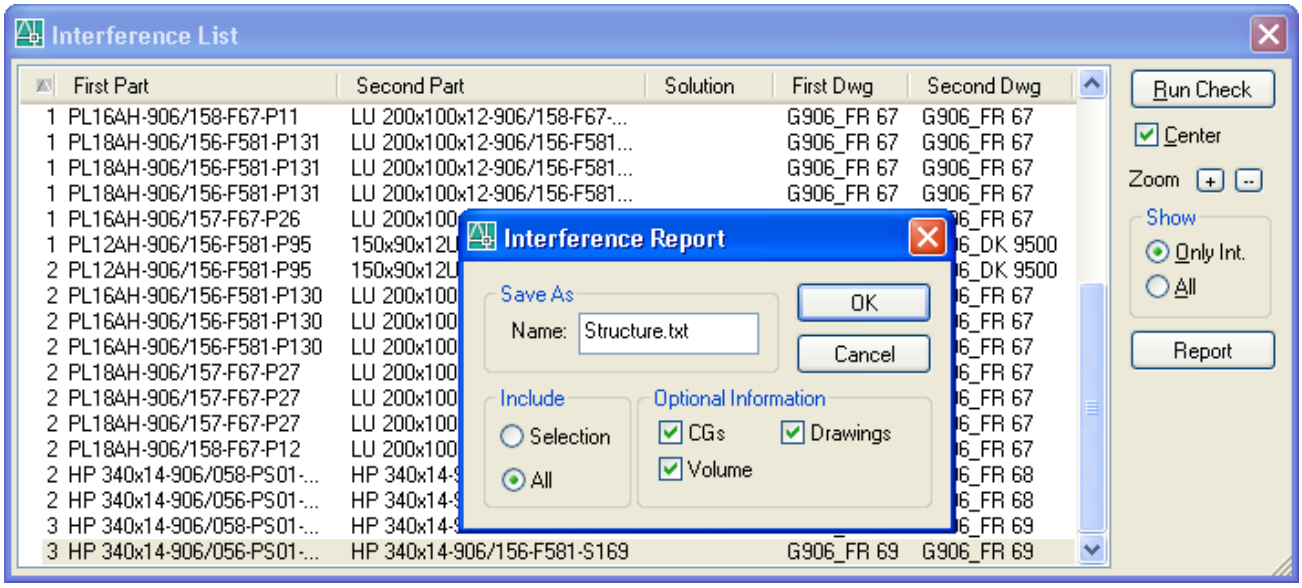


6. To have a better view of an interference in the drawing, turn on Center, right-click on the interference, and choose View. ShipConstructor hides all solids except for those involved in the interference. (You may need to move the Interference List window to see them.) Click the [+] and [--] buttons to zoom in and out.
7. To display the properties of an interference, right-click on it and choose Solution.



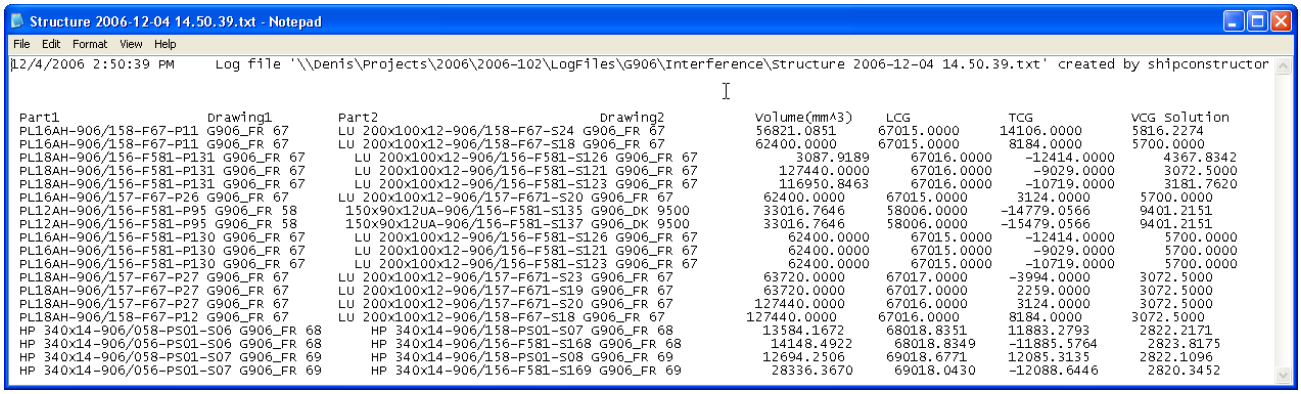
The Solutions window displays the Physical Properties of the interference and lets you enter a suggested Solution. For example: Shorten Stiffener.

8. To generate an interference report, in the Interference List window select the interferences to include in the report and click Report to open the Interference Report window.



9. Enter a Name for the interference report, select the information to include in the report, and click OK.

ShipConstructor displays the report in Notepad and saves the text file in <project folder>\LogFiles\<unit>\Interference\<my interference drawing> <time stamp>.txt.



Check Local Interferences

You can also check for interferences in a single planar group drawing without creating an interference drawing. This will only check for collisions between objects in a single drawing, but it eliminates the additional overhead of creating a new interference drawing.

To Check Local Interferences

1. Open the planar group drawing that you want to check for interferences.
2. Choose SC Utilities > Check Local Interferences (page 466) to open the Interference List window.
3. Follow the steps described above in the Check Interferences section, starting at step 4.

Replicate Objects to Other Planar Groups

You can replicate geometry from the current group drawing to other group drawings that are parallel to the current one. For example, you can replicate from a frame to other frames but not to a deck group.

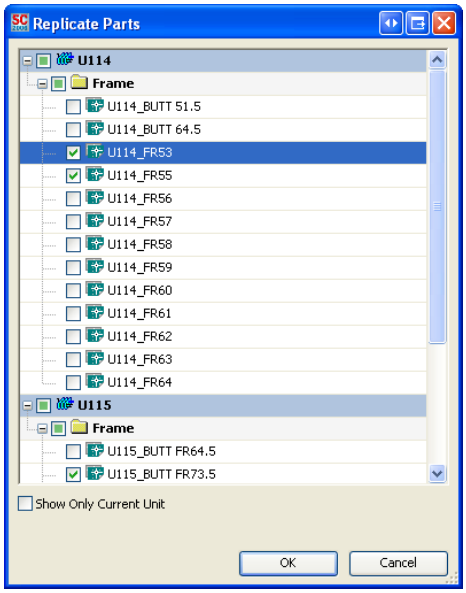
Typically, you would use duplicate objects if significant geometry is identical between several groups. For example, for a frame, the geometry portion below a tank top could be identical for many frames. Only the outside shell plating would change slightly from frame to frame.

To replicate objects to another planar group

1. Choose SC Structure > Replicate Objects to Other Group (page 449)
2. Select the objects or parts that you want to replicate and then press Enter.

> Note: You must save the current drawing in order to continue and you will not be able to undo the replication operation.

The Replicate Geometry window appears.



3. From the tree, choose the groups you want to replicate to. Only groups parallel to the current group appear in the window.
4. Click OK.

ShipConstructor copies the objects you selected and inserts them into the selected planar group drawing. If the objects replicated support identical relationships and the resulting copied object is geometrically identical to the original then an identical relationship is created between the objects if they are within the same unit.

Transfer Objects to another Planar Group

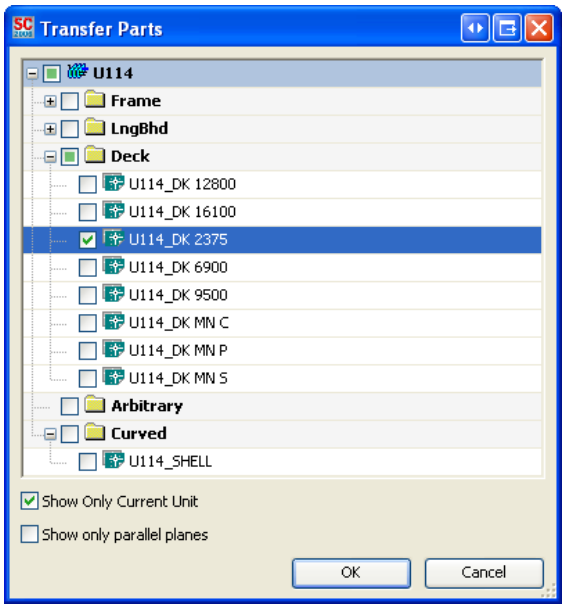
You can transfer geometry from the current group drawing to a different planar group. You are given the option to leave the selected objects in the current drawing which if chosen will make a copy of the selected objects and place them in the selected planar group. None of the transferred objects are moved in relation to the World UCS during the transfer.

To replicate objects to another planar group

1. Choose SC Structure > Transfer Objects to Group (page 449)
2. Select the objects or parts that you want to transfer and then press Enter.

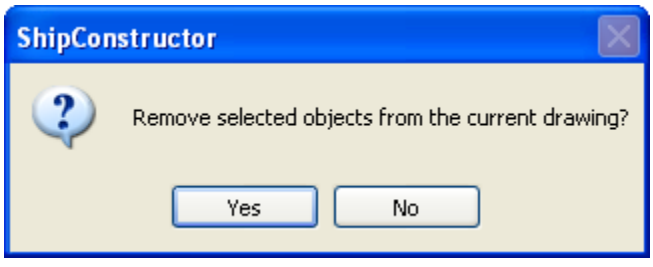
   Note: You must save the current drawing in order to continue and you will not be able to undo the transfer operation.

   The Transfer Parts window appears.



3. From the tree, choose the group you want to transfer to. You can filter the listed planar groups by Current Unit and Parallel Planes.
4. Click OK.

   The confirmation window appears.



5. Select Yes if you want to remove the selected objects from the current drawing. Choosing No will transfer a copy of the selected objects to the chosen planar group.

   ShipConstructor moves\copies the objects you selected and inserts them into the selected planar group drawing. If the objects are copied, they support identical relationships and the resulting copied object is geometrically identical to the original then an identical relationship is created between the objects if they are within the same unit.

Miscellaneous Geometry Tasks

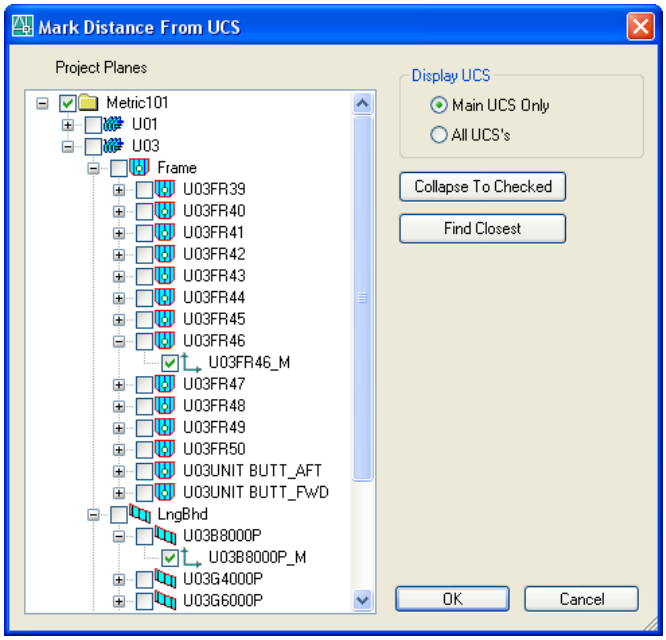
Insert a Label or Dimension From a Point to a Structural Group

You can insert a label or dimension indicating the distance from any point to specified structural groups.

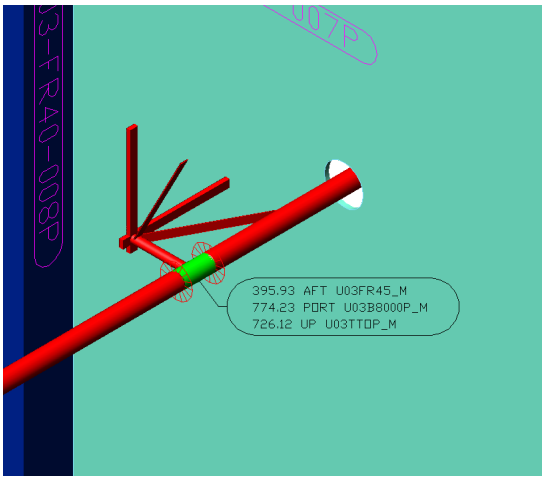
To insert a label or dimension indicating the distance from a point to structural groups

1. Choose SC Assembly > Global Dimension to Point (page 503)
2. Select the point to label or dimension.

   The Mark Distance From UCS window appears.



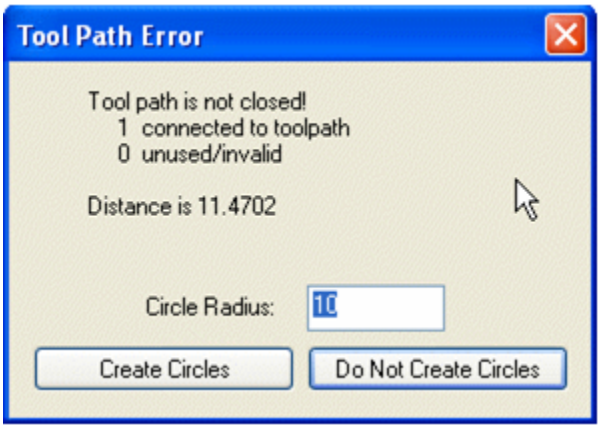
3. Select the structural groups you want from the tree. By default, the nearest groups are automatically selected.
4. Click OK.



Toolpath

The Toolpath command connects objects such as lines, arcs, and polylines to a closed toolpath. This was a required function in ShipConstructor2005, but it is not required in ShipConstructor. The function eliminates duplicate objects and

corrects small gaps. If a gap is discovered that exceeds the maximum snap tolerance (set in Manager > Settings > Preferences > Misc), an error window appears. An error window can look like this:



Click the Create Circles button to create end circles that make it easy to find the problem. For example:

Create a Closed Toolpath

1. Choose SC Utilities > Toolpath (page 460)
2. Draw a window to select all objects outside the toolpath.

   It does not matter if you select objects that are not part of the toolpath. The command only connects objects with no gaps or a gap less than the tolerance specified in Manager. Small gaps are closed if they do not exceed the Snap tolerance specified in Manager.
3. Press Enter.
4. The result of the toolpath operation appears in an information window. Click OK.

Dihedral Angle

Use the Dihedral Angle command to determine the angle between two non-parallel planar objects. The dihedral angle between two planes is the angle between their two normal unit vectors .You can see the dihedral angle of two planes by looking at the planes along their intersection line.

Find the Dihedral Angle between Two Planar Objects

1. Choose SC Utilities > Dihedral Angle (page 461)

Note: ShipConstructor creates the dimension.style SCON_DihedralAngle for displaying dihedral angles in the drawing.

2. Select the two.objects you want to determine the dihedral angle for.
3. Press Enter.
4. ShipConstructor displays the four possible dihedral angles as arcs. Select the dihedral angle that you want to display in the drawing by clicking on its arc.

Convert a 3D Object to a 2D Object

You can convert a 3D object to a 2D object on the XY plane of the current UCS. When you do so, ShipConstructor removes the Z component of the object. For example, a circle that is not parallel to the current UCS is converted to an elliptical polyline.

This may be useful if, for example, a polyline is slightly out of plane, or if you defined objects on the thickness throw UCS and now want them in the construction UCS.

To convert a 3D object to a 2D object

1. Choose SC Utilities > 3D to 2D (page 457)
2. Select the objects to convert.

Project Shapes

You can project a shape from an orthogonal plane (frame, deck, longitudinal bulkhead) to a skewed plane along the X, Y, or Z axis of the world coordinate system.

Note: Projected circles that produce elliptical shapes are converted to line segmented polylines.

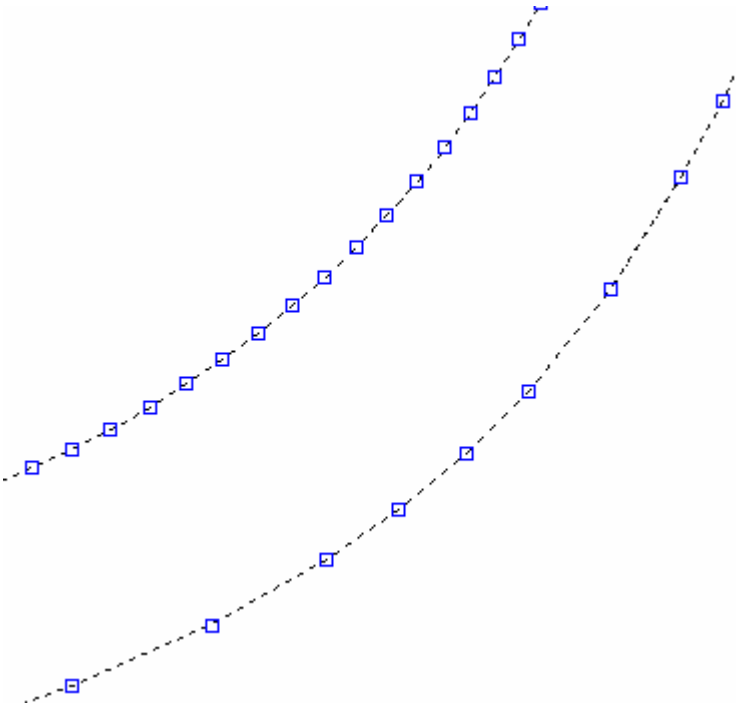
To project shapes from an orthogonal plane to a skewed plane

1. Choose SC Utilities > Orthographic Projection (page 458)
2. Select the objects you want to project.
3. Type the projection direction (in the world coordinate system): X, Y, or Z.

Reduce the Number of Vertices on Polylines

Some polylines may contain a large number of vertices. You can reduce the number of vertices on a polyline by removing vertices that are below a certain tolerance value. You may want to do this for various reasons:

- Display speed – Polylines that contain a large number of vertices take longer to display. Reducing the number of vertices on polylines can speed up your display. This is especially important when displaying solids. (If you create a solid from polylines that contain a large number of vertices, then the solid will also contain a large number of vertices and will slow down your display.)
- Computing speed – Computer calculations, such as interference checking, take longer for solids containing many vertices.
- File size – ACIS solids containing many vertices use enormous amounts of disk space.
- NC machine limitations – Many cutting machines will move erratically or start and stop at each polyline segment if there are too many vertices spaced closely together.



Two identical frame sections. The upper frame section has many vertices (tolerance = 1 mm). The lower frame section has fewer vertices (tolerance = 3 mm).

To reduce the number of vertices on a polyline

1. Choose SC Utilities > Remove Vertices Below Tolerance (page 458)
2. Select the polyline you want to reduce.
3. Press Enter.

```
[No fit arcs] Tolerance <0.05>:
```

4. To change the fit arcs option, type N for no fit arcs or F for fit arcs and press Enter.
5. Enter a tolerance value.

   Tip: In general, a 0.1 mm (1/32") tolerance is a good compromise between speed and accuracy.

6. Press Enter.

Convert an Ellipse or Spline to a Polyline

NC machines do not understand ellipses or splines. You must convert them to polylines.

To convert an ellipse or spline to a polyline

1. Choose SC Utilities > Convert Ellipse/Spline to Polyline (page 459)

   The default tolerance appears in the command line.
2. To change the tolerance, type T and press Enter. Then type the tolerance and press Enter.

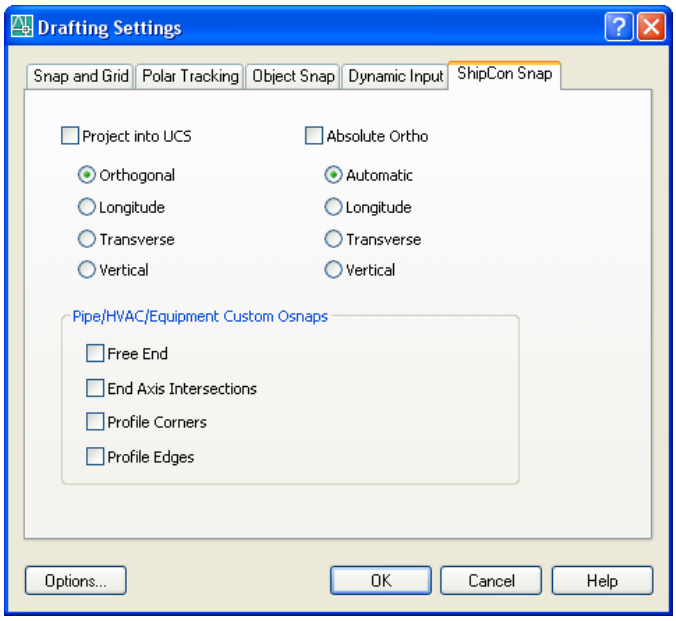
   A good tolerance value is about 0.1 mm (1/32").
3. Select the ellipse or spline you want to convert.
4. Press Enter.

Set Up Snapping

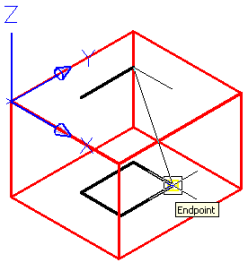
ShipConstructor includes several snap settings that make it easier to work in three dimensions.

To set up snapping

1. Choose SC Utilities > Snap (page 467) to open the Drafting Settings window.



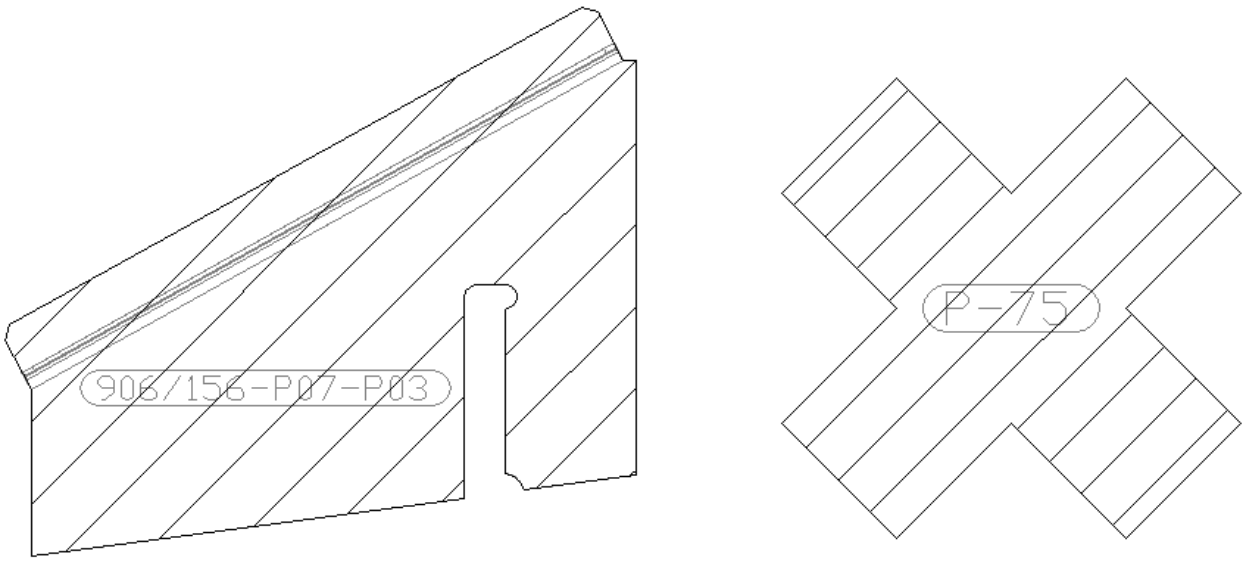
2. Set the options.
   - Project into UCS – Projects the picked point into the current UCS. For example, when drawing a line that you want in the UCS, you can pick points at elevation and they will be projected down onto the UCS. You can choose to project perpendicular to the plane (Orthogonal), along the WCS X axis (Longitude), along the WCS Y axis (Transverse), or along the WCS Z axis (Vertical).



   - Absolute Ortho – Lets you draw orthogonal about the world coordinate system, independent of the current UCS. (The AutoCAD **ORTHO** command only lets you draw orthogonal to the current UCS.)
   - Free End – See the HVAC manual for details.
   - End Axis Intersections – See the HVAC manual for details.
   - Profile Corners – See the HVAC manual for details.
   - Profile Edges – See the HVAC manual for details.
3. Click OK.

Invalid Parts

If there are any problems creating or drawing a part, that part may become invalid. Invalid parts are usually displayed hatched, as shown below:



Invalid parts - if a valid boundary is not found the part is displayed as an X.

A part can become invalid for many reasons: the boundary is not closed, problems with the geometry or solid creation, or corrupt data in the database, for example. It is not always possible to determine the exact cause. The main purpose of invalid parts is to provide a mechanism to delete parts from the database that may otherwise not show up in the drawing. Deleting and re-creating the part is usually the best solution, but there are some common problems that may be fixed. These solutions may not be applicable to all part types:

- Check that the piecemark is inside the part's boundary. If not, move it back inside with the grip points.
- Check that the part's boundary is closed. If not, adjust the part's boundary objects or add boundary lines to close any gaps.
- If a curved plate solid is created from a Rhino file, its production geometry will be shown as invalid. (See Curved Plate Creation (page 184)).

If the part remains invalid then the only other option is to delete and re-create the part. Also note that any commands run on the invalid part may not function as expected.

Check Structure

Check Product Hierarchy

Checking the product hierarchy is not an automated command. You need to check the assignment of parts to assemblies. There can be cases where a modeler has mis-assigned a part to the wrong assembly or the assembly stages are not completely known at the time of modeling.

To check an assembly

1. Create a product hierarchy drawing.
2. Check for correct assignment of parts to assemblies.

To check assemblies

1. Create assembly drawings for the unit.
2. Open each assembly drawing and verify that all the parts are there.

Check a Unit

When you check a unit, ShipConstructor checks the complete unit and its structure model drawings for errors. Before the check is run, you may choose to repair errors. If you select No, no modifications will be done to the drawings and errors will only be reported in the log file.

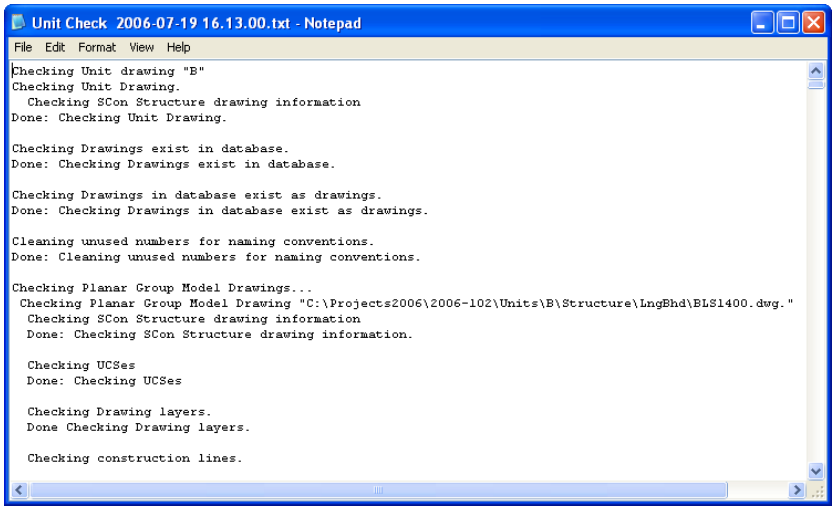
The unit check also updates the drawings to reflect any standards changes (if the option to repair errors was selected). When a change in the standards would change the physical properties of a part, the drawings need to be opened and saved to update weight and center of gravity data.

Checking a unit will also check that template drawings used by any of those unit's drawings are both registered to the project and the drawing file exists. If the option to repair errors is selected, missing drawing files will be removed from the project, and any drawings that are not registered will be. Currently, the template types checked are Assembly, Pipe/HVAC/Equipment Arrangement, and Pipe/HVAC Spool.

To check a unit drawing

1. Choose ShipConstructor > Check > Check Unit (page 384)

   ShipConstructor checks the unit drawing and displays the following message when finished; including a log file that contains a summary of the check.


Unit Check 2006-07-19 16.13.00.txt - Notepad
File Edit Format View Help
Checking Unit drawing "B"
Checking Unit Drawing.
Checking SCon Structure drawing information
Done: Checking Unit Drawing.
Checking Drawings exist in database.
Done: Checking Drawings exist in database.
Checking Drawings in database exist as drawings.
Done: Checking Drawings in database exist as drawings.
Cleaning unused numbers for naming conventions.
Done: Cleaning unused numbers for naming conventions.
Checking Planar Group Model Drawings...
Checking Planar Group Model Drawing "C:\Projects2006\2006-102\Units\B\Structure\LngBhd\BLS1400.dwg."
Checking SCon Structure drawing information
Done: Checking SCon Structure drawing information.
Checking UCSes
Done: Checking UCSes
Checking Drawing layers.
Done Checking Drawing layers.
Checking construction lines.

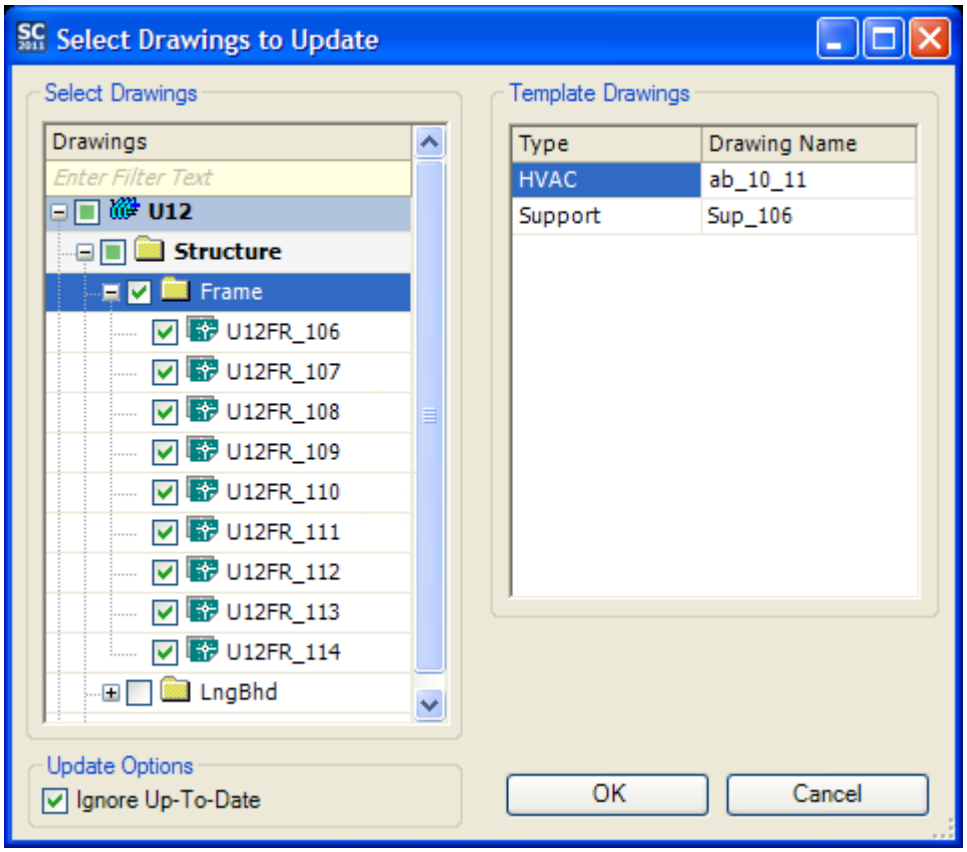
Note: This log file can be rather long. Any errors and warnings can be easily found by searching (using Notepad's Edit > Find) for either "Error" or "Warning".

Update Model and System Drawings

You can use the Update Model and System Drawings command to recreate drawings in your project, or update them if they are out-of-date, using information from the database. This is especially useful if you do not have the permissions required to edit or create drawings, since otherwise the drawing would have to update itself every time it is opened.

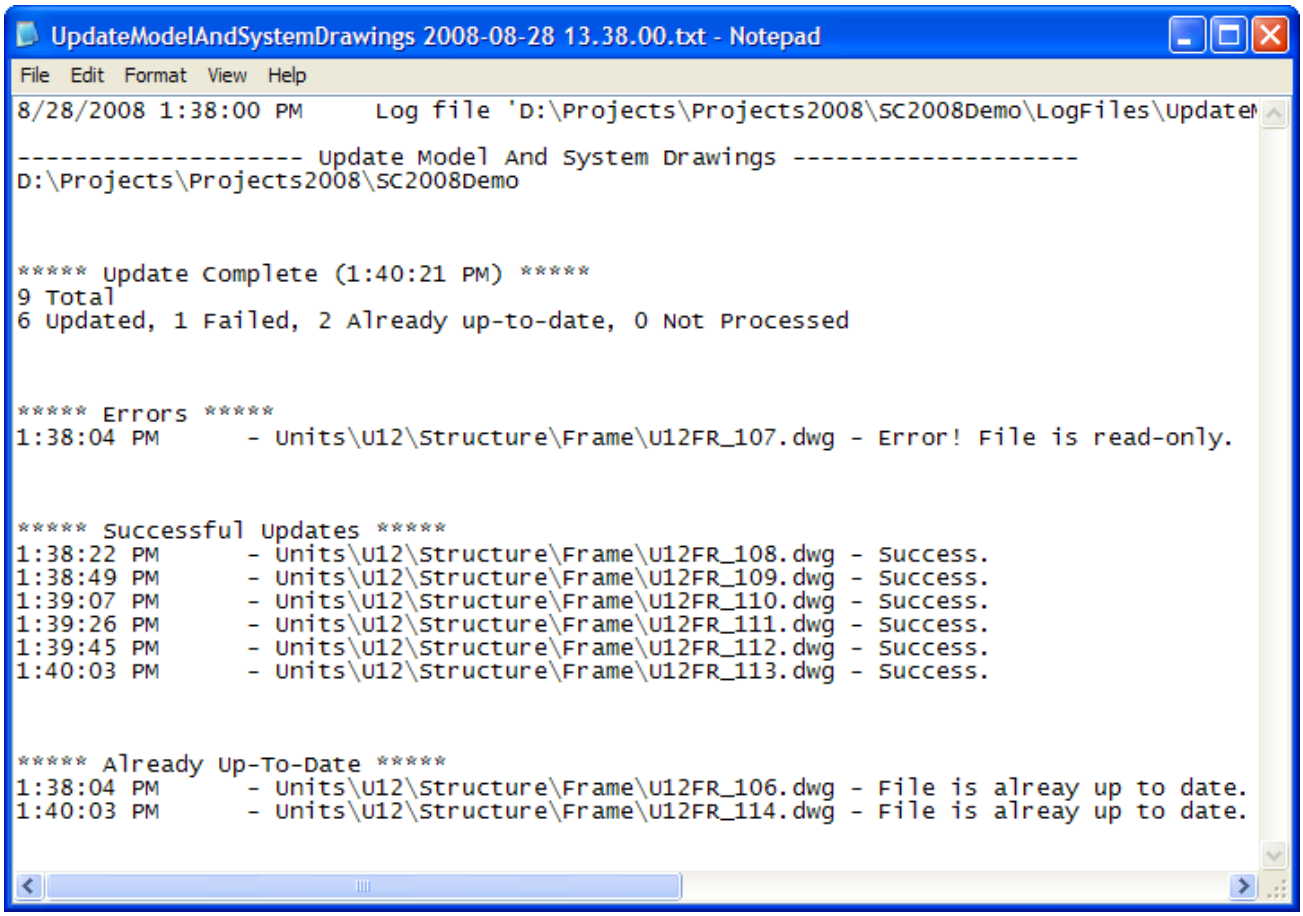
To update model and/or system drawings

1. Choose ShipConstructor > Update Model and System Drawings (page 385).
2. The Select Drawings to Update window appears.



Select the drawings you wish to recreate or update using the ShipConstructor database. Optionally you may check the Ignore Up-To-Date box, which will improve the speed of the command by not updating files that do not require updating from the database.

3. Click the OK button. ShipConstructor will open and update each of the selected drawings.
4. When the operation is complete, a status message is displayed along with the option to open a log file containing a detailed breakdown of the status for each selected file.

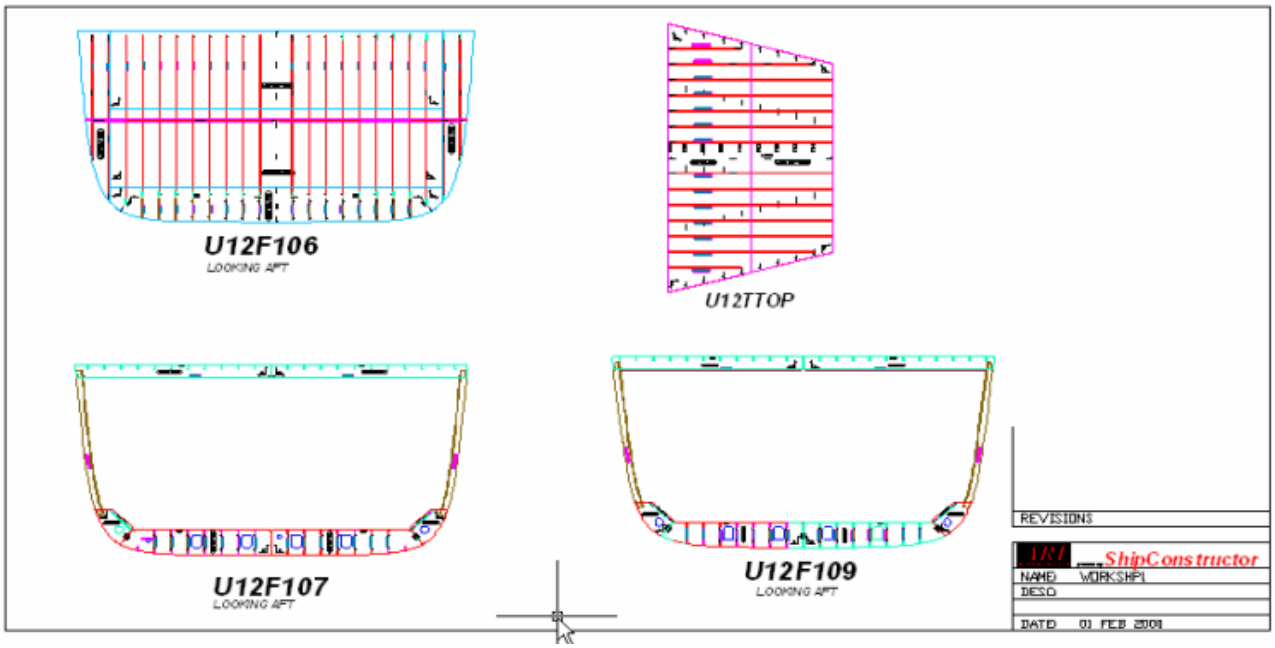

UpdateModelAndSystemDrawings 2008-08-28 13.38.00.txt - Notepad
File Edit Format View Help
8/28/2008 1:38:00 PM Log file 'D:\Projects\Projects2008\SC2008Demo\LogFiles\UpdateM
---------- Update Model And System Drawings ----------
D:\Projects\Projects2008\SC2008Demo
***** Update Complete (1:40:21 PM) *****
9 Total
6 Updated, 1 Failed, 2 Already up-to-date, 0 Not Processed
***** Errors *****
1:38:04 PM - Units\U12\Structure\Frame\U12FR_107.dwg - Error! File is read-only.
***** Successful Updates *****
1:38:22 PM - Units\U12\Structure\Frame\U12FR_108.dwg - Success.
1:38:49 PM - Units\U12\Structure\Frame\U12FR_109.dwg - Success.
1:39:07 PM - Units\U12\Structure\Frame\U12FR_110.dwg - Success.
1:39:26 PM - Units\U12\Structure\Frame\U12FR_111.dwg - Success.
1:39:45 PM - Units\U12\Structure\Frame\U12FR_112.dwg - Success.
1:40:03 PM - Units\U12\Structure\Frame\U12FR_113.dwg - Success.
***** Already Up-To-Date *****
1:38:04 PM - Units\U12\Structure\Frame\U12FR_106.dwg - File is alreay up to date.
1:40:03 PM - Units\U12\Structure\Frame\U12FR_114.dwg - File is alreay up to date.

Note: You can also find the log file in the LogFiles folder contained in your project's main folder.

Approval Drawings

An approval drawing is a two-dimensional view of the model. Approval drawings are used as general arrangement drawings or classification drawings (for example, for initial design and classification).

Note: Do not use approval drawings for production. Instead, use assembly drawings.



Approval drawing

Create an Approval Drawing Template

Approval drawing templates control the appearance of approval drawings. ShipConstructor includes a default approval drawing template.

To create an approval drawing template

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.
2. Select the Templates page.
3. Select the Approval folder.
4. Click New Approval.

   The New Drawing window appears.
5. Enter a name for the drawing.
6. Click OK.

Create an Approval Drawing

To create an approval drawing

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.
2. Select the Approval page.

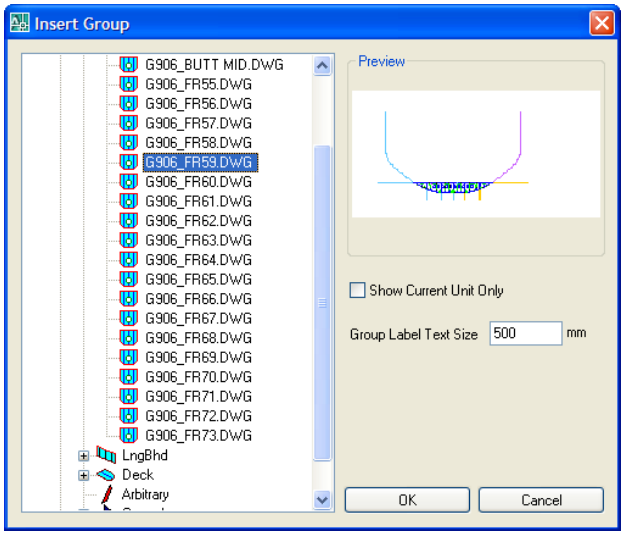
3.

4. Click New to open the New Drawing window.
5. Enter a name for the drawing.
6. Click OK.

Insert a Planar Group into an Approval Drawing

To insert a planar group into an approval drawing

1. Choose SC Approval > Insert Group (page 488) to open the Insert Group window.



2. Select the planar groups you want to insert.
3. Click OK.

ShipConstructor inserts the planar group drawing into the approval drawing and starts the move command.

Note: If you are not using an approval drawing template, your view may be zoomed in too close to see the entire planar group.

4. Move the planar group to the location you want.

Note: Planar groups are inserted into approval drawings as Xrefs. Whenever a planar group drawing changes, ShipConstructor automatically updates the approval drawing. If you want to add dimensions to an approval drawing, it is usually easier to add the dimensions to the planar group drawing on a special layer and then show that layer in the approval drawing.

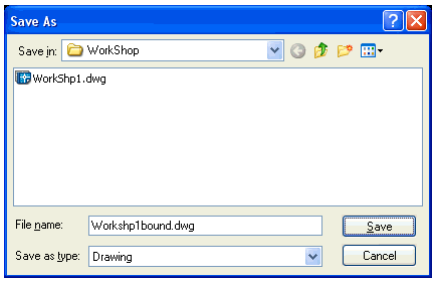
Save a Bound Approval Drawing

Planar groups and parts are inserted into approval drawings as Xrefs. You can save an approval drawing as a bound approval drawing that contains the actual planar groups and parts (that is, it does not contain Xrefs to other drawings). If a planar group or part changes, ShipConstructor will not automatically update a bound approval drawing.

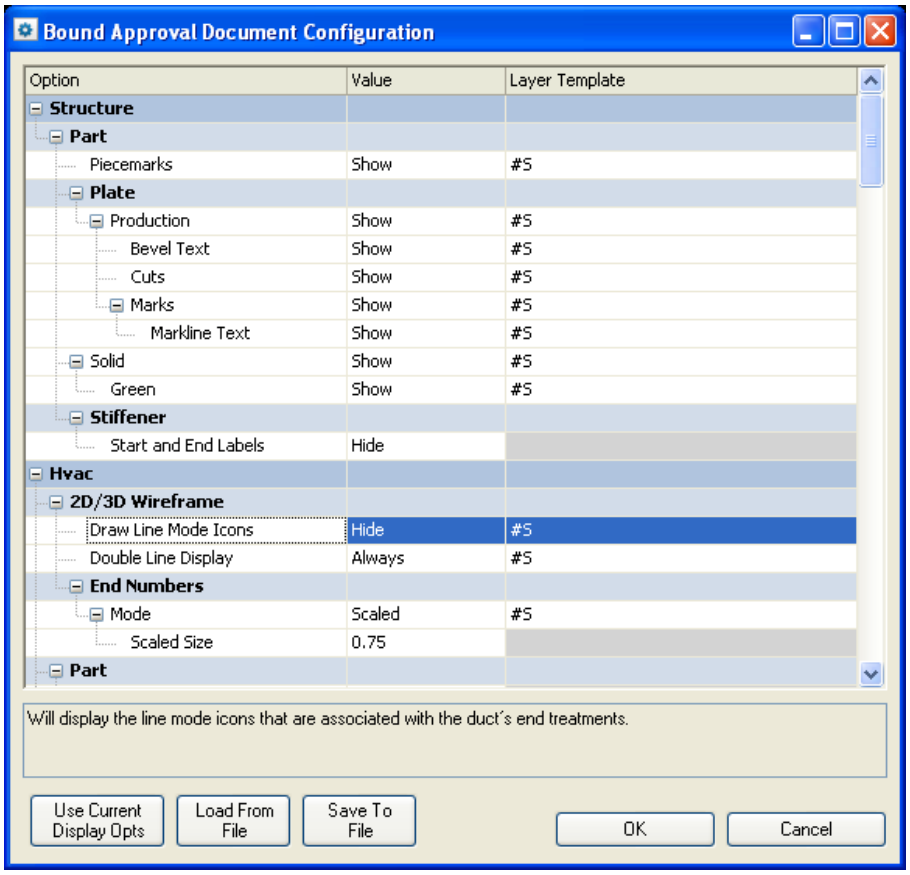
To save a bound approval drawing

1. Choose SC Approval > Save As Bound Approval (page 489)

   The Save As window appears.



2. Enter a new File name. Once the file name has been selected the Bound Approval Document Configuration window will appear.

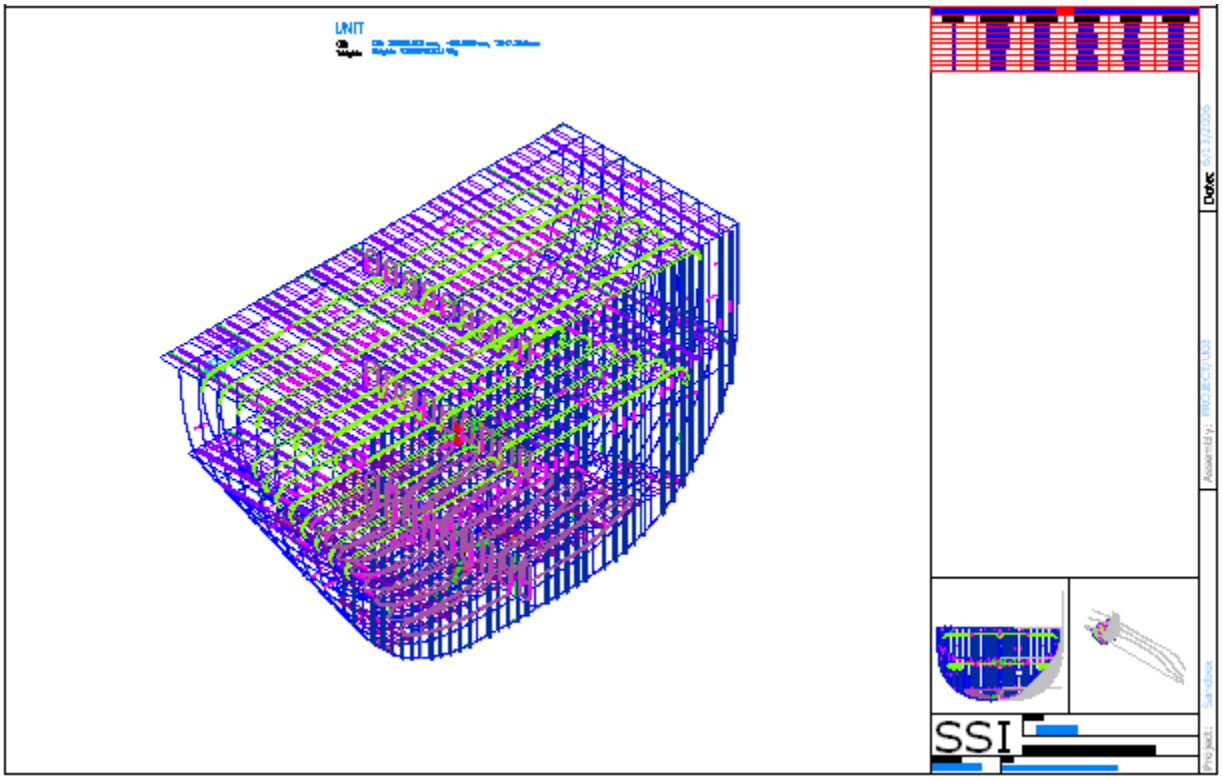


For Details on configuration using this interface see Export To Dwg

3. Once you are done configuration select OK and the Bound Approval Document will be created.

Assembly Drawings

An assembly drawing shows all parts within an assembly. Each part may be annotated, making it easy for production to fabricate the assembly.



Before creating an assembly drawing, make sure you have done the following:

- Created a naming convention for assembly drawings. See Naming Conventions (page 35).
- At least one assembly drawing template.

Set Up Assembly Drawings

Before creating assembly drawings, you must set up the assembly drawing template.

Set Up an Assembly Drawing Template

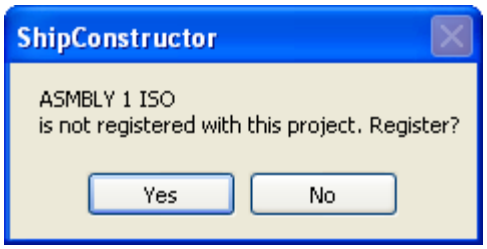
You can control the layout of assembly drawings using assembly drawing templates. ShipConstructor includes one default assembly drawing template that you can use or modify. You can also update your existing ShipConstructor2005 assembly drawing templates, or you can create new assembly drawing templates.

Note: There must be at least one assembly drawing template defined before you can create an assembly drawing.

Update a Previous Version of an Assembly Drawing Template

To update an existing ShipConstructor 2005 assembly drawing template

1. Make sure the template you want to update is in the current project path.
2. Choose ShipConstructor > Navigator (page 368) to open the Navigator.
3. Select the Templates page.
4. Select the assembly template drawing you want to update.
5. Click Open.



6. Click OK to register this template with the current project.
7. Choose SC Assembly Template > Update Template drawing.

   Note: Some keywords' text may change as they are converted to new ShipConstructor keyword entities.

   If there was a <BOM> keyword in the existing template, it will not be updated. You must insert an empty BOM into the template drawing manually.

Create an Assembly Drawing Template

To create an assembly drawing template

1. Choose ShipConstructor > Navigator to open the Navigator.
2. Select the Templates page.
3. Select the Assembly folder.
4. Click New Assembly to open the New Drawing window.
5. Enter a name for the drawing and click OK.
6. Insert your company title block, or create one.
7. Set up as many viewports as you need.
8. Move and size each viewport individually.
9. Double-click on a viewport and choose View > Shade to select a shading style for the assembly in the view.

   See Set Up Assembly Drawing Template Viewports (page 220) and Insert Keywords into an Assembly Drawing Template (page 221).

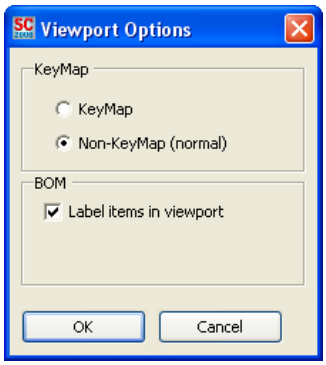
Set Up Assembly Drawing Template Viewports

Each viewport in an assembly drawing template can either be a keymap viewport or a normal viewport. A keymap viewport displays the assembly parts and the unit, letting you see the location of the assembly within the unit. A normal viewport displays only the assembly.

Each viewport (keymap or normal) in an assembly drawing can also contain automatic annotations. When you indicate that you want a viewport to contain automatic annotations, ShipConstructor inserts annotation bubbles to automatically label the parts that are visible in the viewport.

To set up assembly drawing template viewports

1. In the assembly drawing template, activate paper space.
2. Click in the viewports that you want to set up.
3. Choose SC Assembly > Viewport Options (page 493) to open the Viewport Options window.



4. To make the viewport a keymap or normal viewport, choose Keymap or Non-Keymap (normal).
5. To make the viewport contain automatic annotations, choose Label items in viewport.
6. Click OK.
7. Repeat Steps 2 through 6 for every viewport you want to set up.

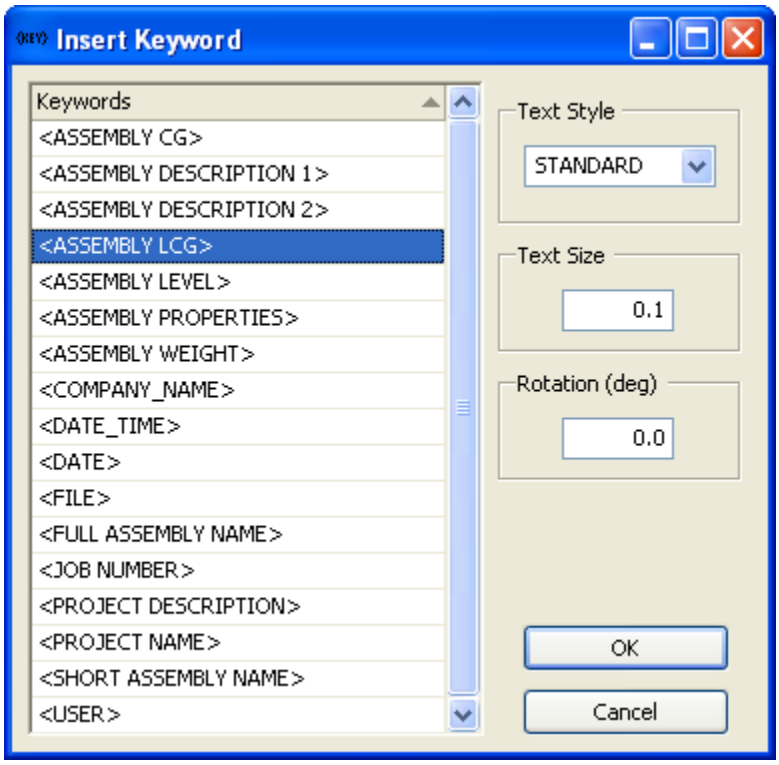
Insert Keywords into an Assembly Drawing Template

In ShipConstructor2005, keywords were text entities. In ShipConstructor, keywords are ShipConstructor keyword objects.

You can insert various keywords into an assembly drawing template. When you generate an assembly drawing, ShipConstructor automatically replaces the keyword with the appropriate information.

To insert keywords into an assembly drawing template

1. In the assembly drawing template, activate paper space.
2. Choose SC Assembly Template > Insert Keywords (page 505) to open the Insert Keyword window.



3. Select the keywords to insert.
4. Change the text properties now in this window or later using AutoCAD. Keywords are based on AutoCAD text objects so all the properties of text objects are available.

5. Click OK.
6. Select the location for the keyword.

   The keyword is created as a new ShipConstructor keyword object.

> Note: To change the prefix or postfix text of the keyword, use AutoCAD's property window. When a keyword is updated, the prefix and postfix text will not change.

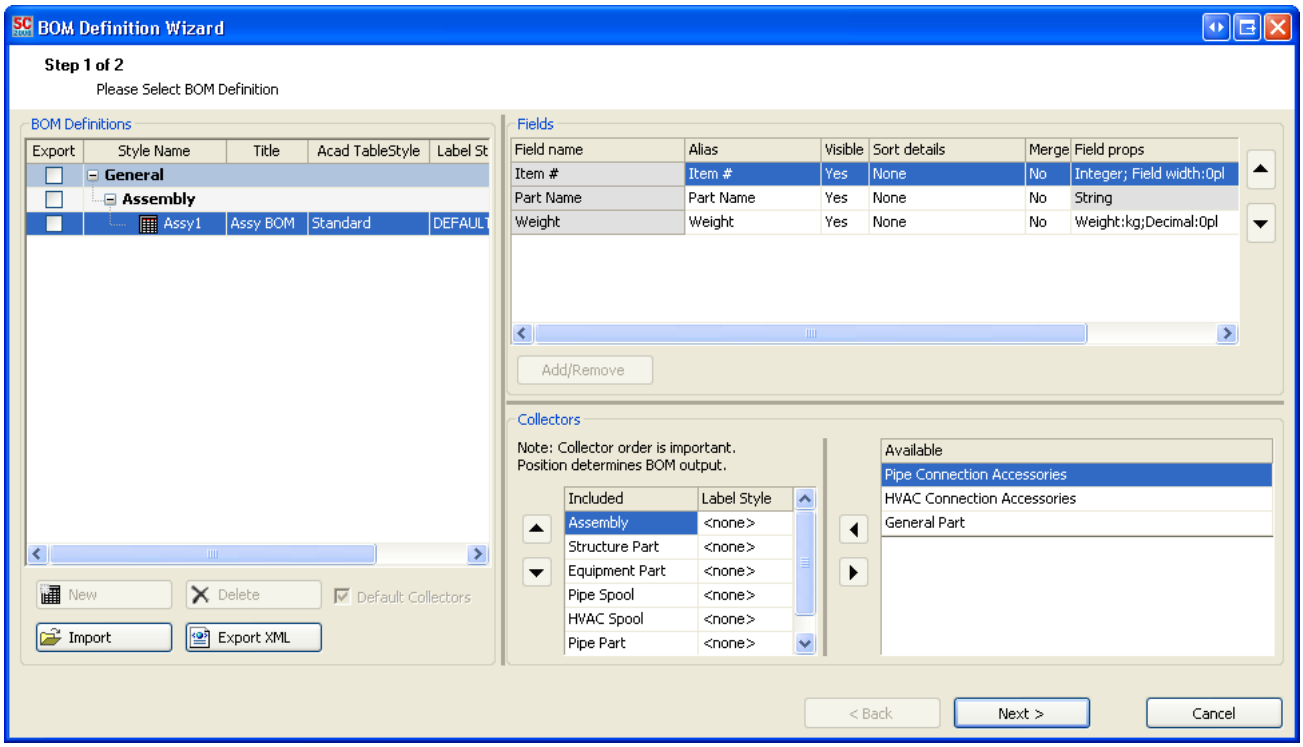
Insert a BOM Table into an Assembly Drawing Template

You can now predefine a BOM definition for use within an assembly drawing template. This enables you to format an empty BOM table to your specifications (correct text style, size, and so on) before creating an assembly drawing.

> Note: To insert the BOM table, there must already be an assembly BOM definition defined. See Set Up an Assembly Drawing BOM Style (page 223).

To insert a BOM table

1. In the assembly drawing template, activate paper space.
2. Choose Format > Table Styles to customize the table style to be used for the BOM table. Be sure that the name of the table style matches the Acad TableStyle name in the BOM Definition.
3. Choose SC Assembly Template > Insert BOM Table... (page 507)



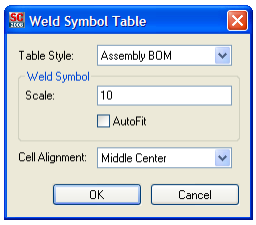
4. Select an Assembly BOM.
5. Click Next.
6. Enter the options for the BOM table.
7. Click Finish.
8. Select the location of the empty BOM.

Insert a Weld Symbol Table into an Assembly Drawing Template

You can insert a table object into the template drawing, which will contain all the weld symbols and their corresponding scenarios. The table in the template drawing is a simple 1x2 empty table that will automatically be filled in during the creation/update of the assembly drawing.

To insert a Weld Symbol Table

1. In the assembly drawing template, activate paper space.
2. Choose SC Assembly Template > Insert Weld Symbol Table (page 507)



3. Select a table style, scale for the Weld Symbols, and an aligment for the table's cells.
4. Click OK.
5. Select a location for the empty table.

Set Up an Assembly Drawing BOM Style

You can set up any number of BOM styles for use in assembly drawings, nest drawings, arrangement drawings, and spool drawings.

To set up a BOM style

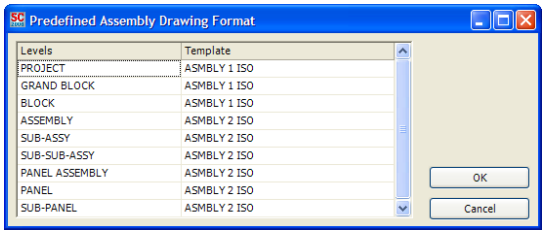
1. In Manager, choose General > Production Output > Bill of Materials (page 339) to open the BOM Definitions window.
2. To create a new assembly BOM style, click on Assembly in the left list and then click New.
3. Click Add/Remove to add fields to the BOM. See BOM Definitions Manager (page 47) for more details on BOM setup.
4. Include any relevant collectors. Make sure the collectors you want are listed under the Included list. See Collector Functionality (page 341) for more about collectors.
5. Choose a label style for each collector. This is the style used to automatically annotate the drawing if this option is selected.
6. Click OK.

Select Predefined Templates

When creating an assembly drawing, you can choose to use predefined templates. When this option is selected, each assembly will use the template assigned here.

To set up predefined templates

1. In Manager, choose General > Production Output > Predefined Assembly Format (page 342) to open the Predefined Assembly Drawing Format window.



2. For each assembly level (Levels), select a template to use for assembly drawings (Template Drawing). (To set up an assembly drawing template, see Create An Assembly Drawing Template (page 220).)
3. Click OK.

Generate Assembly Drawings

Set Up the Keymap Drawing

The keymap drawing is used in keymap (or keyplan) viewports in assembly drawings. The keymap drawing usually contains a wireframe model of the whole unit or vessel. When you create assembly drawings, ShipConstructor can automatically insert (xref) the keymap drawing into the keymap viewports (see Set Up Assembly Drawing Template Viewports (page 220)). The assembly drawing's objects are highlighted within the keymap viewport using color, so you know where that assembly is positioned within the unit.

The contents of the keymap drawing are completely up to you. However, ShipConstructor does populate the keymap with the HullTrace construction lines of the current unit (and their related construction lines) when Update Keymap is clicked. Place all the objects in the keymap drawing on the Keymap layer.

The keymap viewports within each assembly drawing are linked to the keymap drawing. If you generate assembly drawings and later update or modify the keymap drawing, ShipConstructor automatically updates the keymap within all assembly drawings.

To create or regenerate the keymap drawing

> Note: You cannot update the keymap drawing if you have it open.

1. Choose ShipConstructor > Navigator (page 368) to open the Navigator.
2. Select the Assembly page.
3. Click Update Keymap.

ShipConstructor updates the keymap drawing by:

- Deleting any existing KEYMAP drawing.
- Creating a new KEYMAP drawing.
- Copying all the HullTrace construction lines for the whole unit and placing them on the Keymap layer.

To modify the keymap drawing

1. Choose ShipConstructor > Navigator (page 368) to open the Navigator.
2. Select the Assembly page.
3. Select the KEYMAP drawing.
4. Click Open.
5. Make the modifications you want and save the drawing.

All assembly drawings will show the modified keymap drawing.

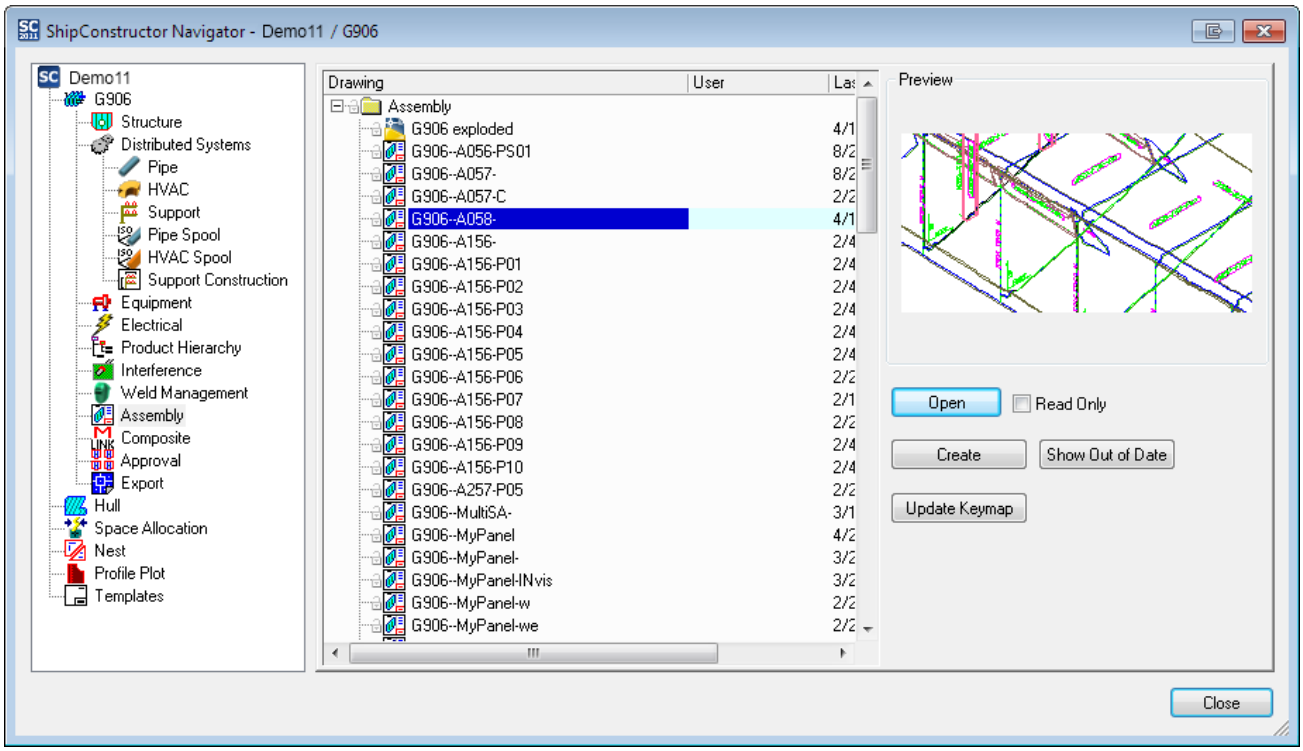
Generating Assembly Drawings

It is usually easiest to create all assembly drawings for a unit or stage at once; however, you can create assembly drawings for individual assemblies if desired.

Once you have met the requirements outlined in Assembly Drawings (page 218), you can create your assembly drawing.

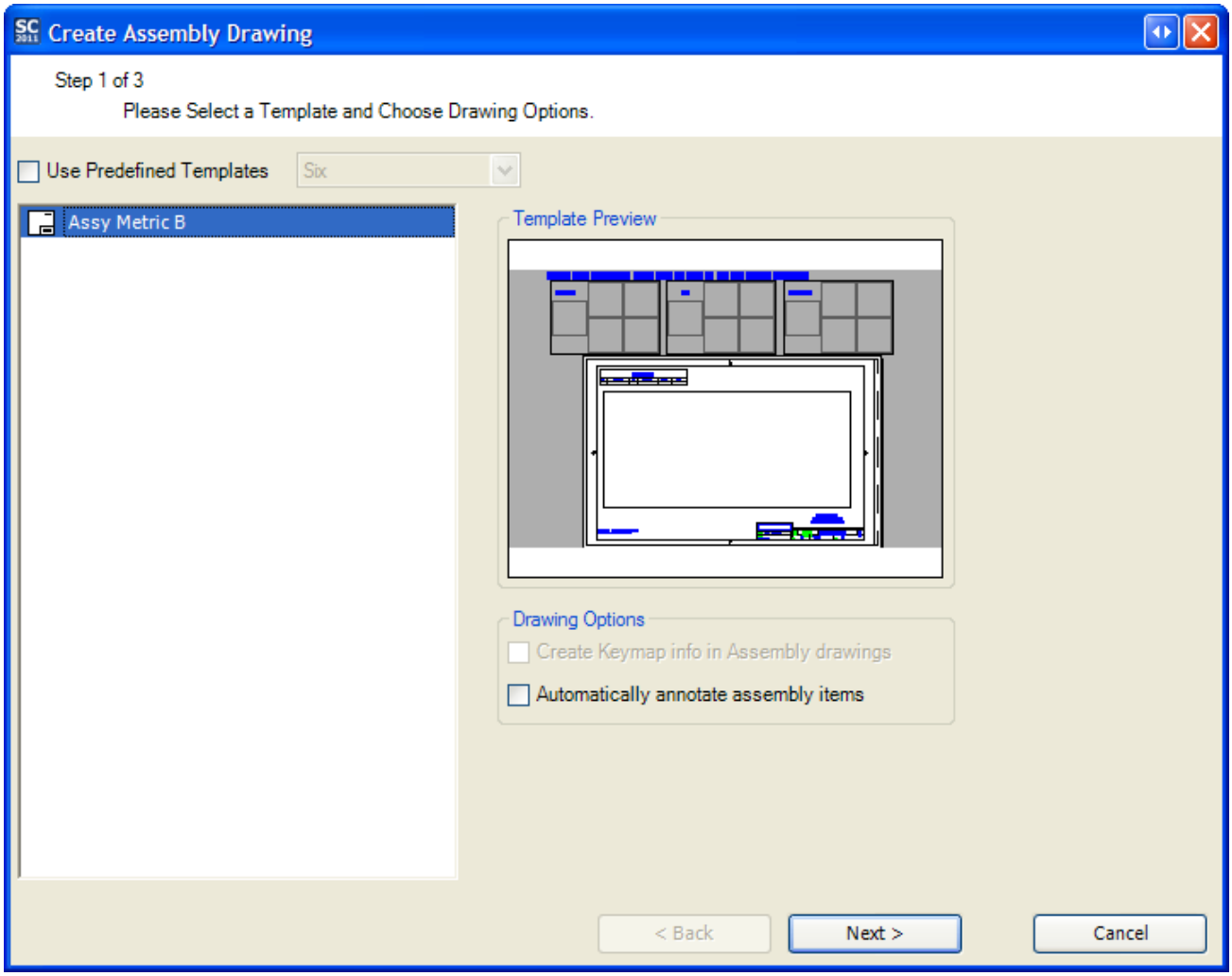
To create assembly drawings

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.
2. Select the Assembly page.



3. Click Create.

   The Create Assembly Drawing wizard appears.



4. Select an assembly drawing template for all your drawings to use, or click the check box to use the predefined assembly templates for each assembly level selected.

   Note: The Use Predefined Templates check box can only be selected if you have set up predefined assembly drawing formats in Manager. See Select Predefined Templates

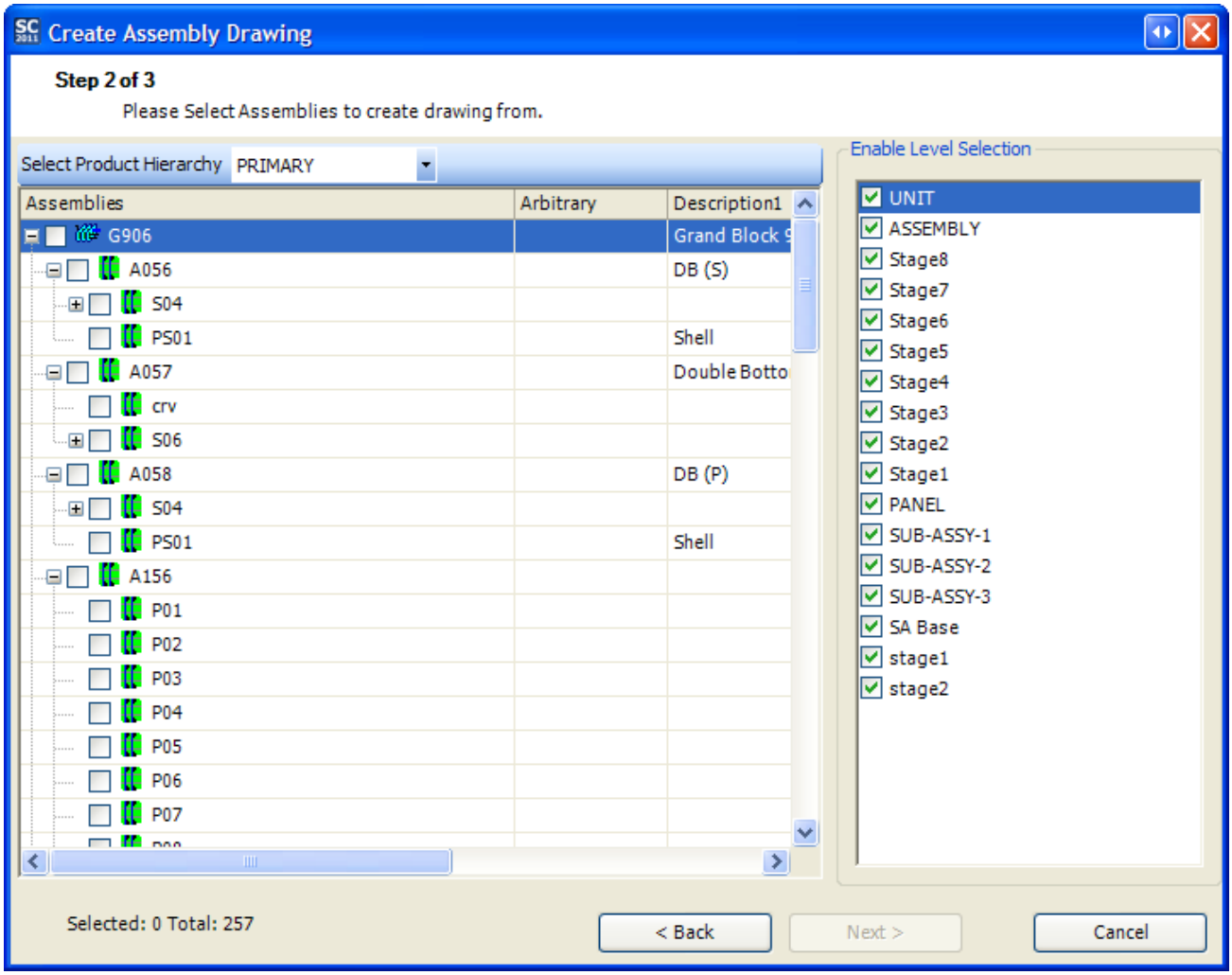
5. Choose assembly drawing options.

   - To include keymaps within the assembly drawings, check the Create Keymap info in Assembly drawings check box.

   Note: The Create Keymap info check box will only be selectable if there was a keymap generated.

   - To automatically annotate the drawing, check the **Automatically annotate assembly items** check box.

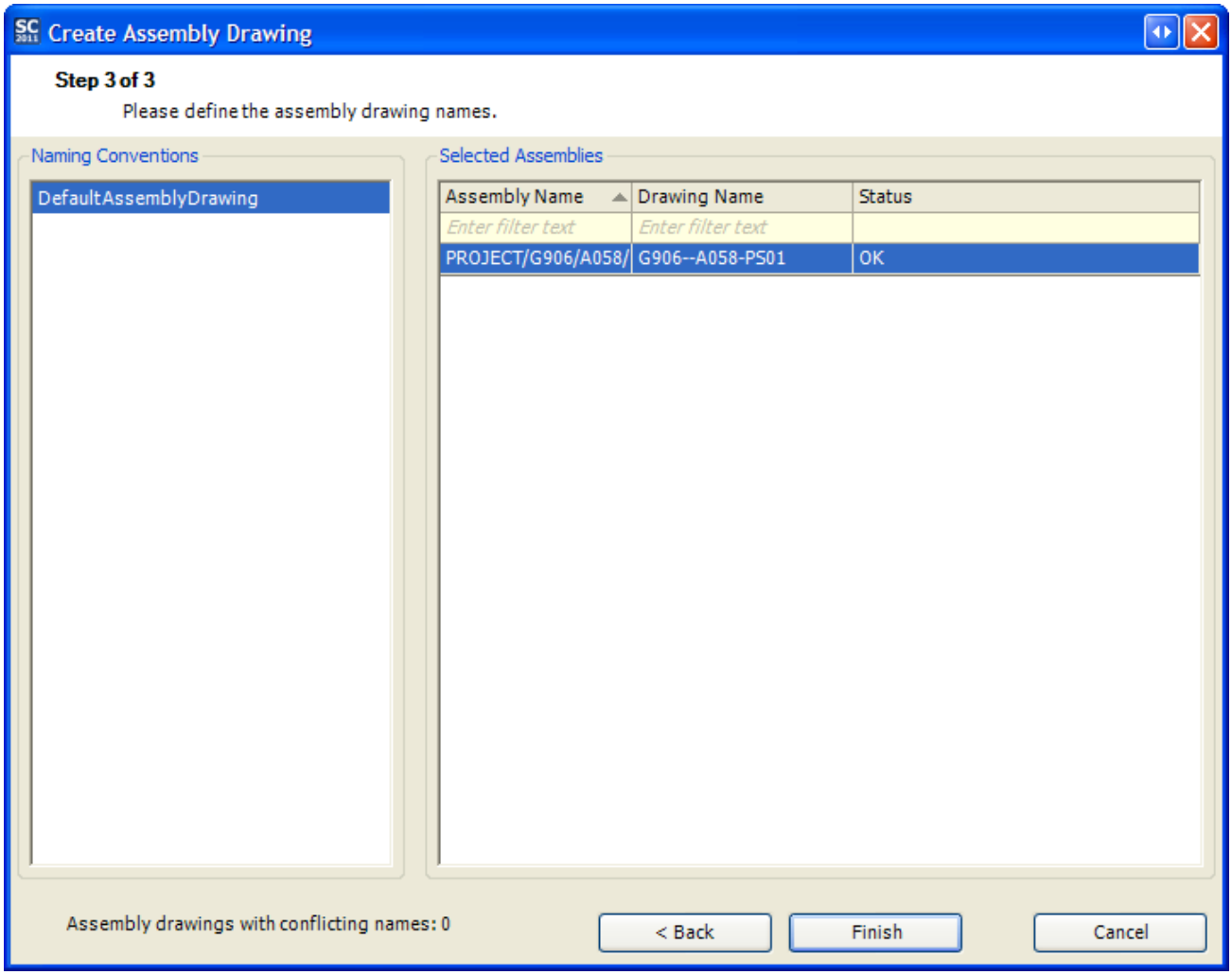
6. Click Next. Step 2 of the Create Assembly Drawing Wizard appears.



7. Place a check in the check box next to each assembly that will become an assembly drawing.

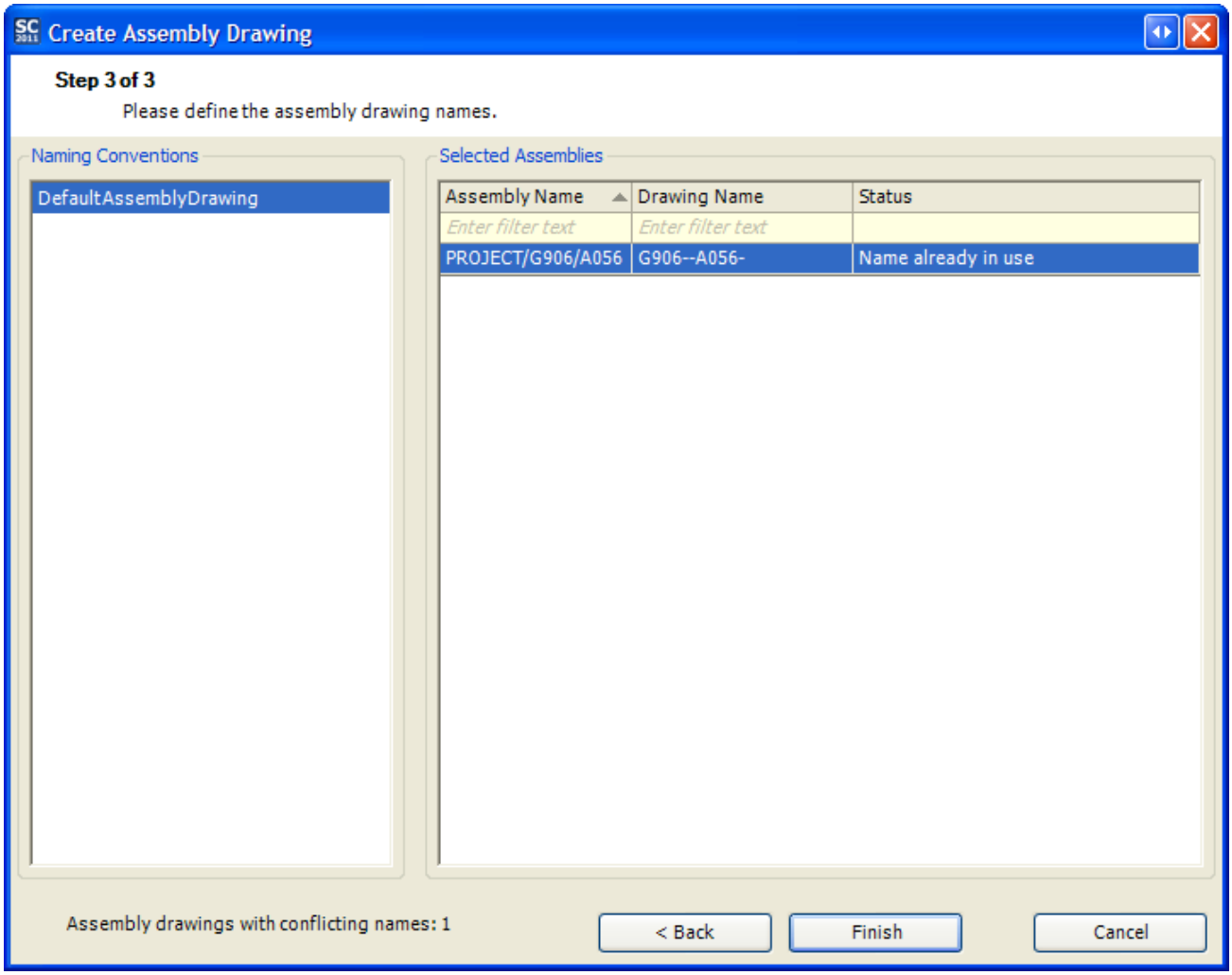
   Tip: By unchecking a level in the Enable Level Selection list, the corresponding assemblies are grayed out and will not be checked. This feature can help speed up selection of your assemblies.

8. Click Next. Step 3 of the Create Assembly Drawing Wizard appears.



Drawing names are previewed for each assembly selected on the previous page. Changing the naming convention will update the preview name. If any of the generated names are currently in use by an existing assembly drawing, the status is changed to Drawing already exists.

If any of the selected assemblies will generate the same name, then the status is shown as Name already in use. Editing the preview name of either conflicting assembly will clear the conflict.



9. Click Finish.

   ShipConstructor creates one assembly drawing for each assembly selected.

   If you are in SDI mode, ShipConstructor will open the last assembly drawing it creates. Otherwise, all assembly drawings created will be open.

Edit Assembly Drawings

The commands for editing assembly drawings are located in the SC Assembly Menu (page 489). Some commands are also available from the Assembly Toolbar (page 477).

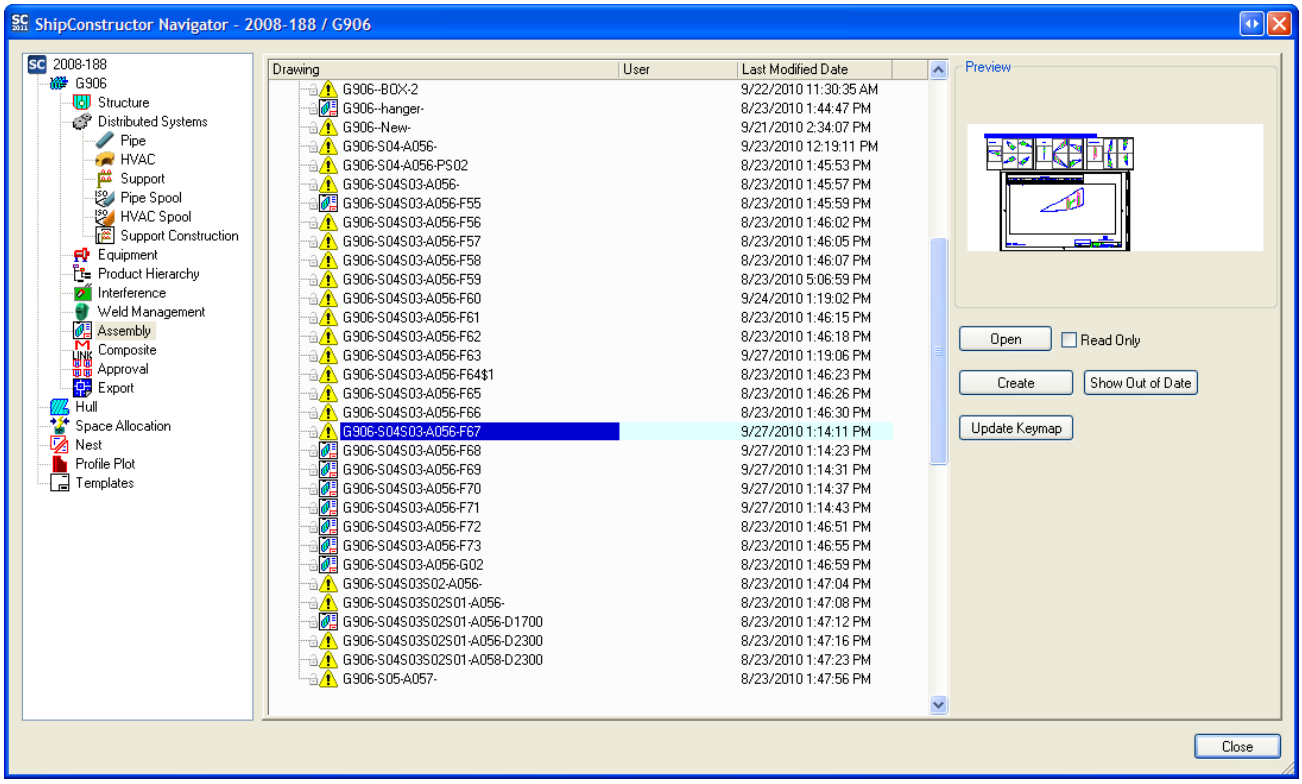
Update Parts in an Assembly Drawing

If changes are made to parts in an assembly after you have created assembly drawings, you must update your assembly drawings (for example, if someone adds a final pipe penetration to a part in the structural model). When you update an assembly drawing, ShipConstructor automatically updates the parts, keywords, and weld symbol tables within the drawing, leaving all of your manual edits and annotations intact. Updating assembly drawings is much faster than re-creating them.

To see which assembly drawings need updating

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.

2. Select the Assembly page.
3. Click Show Out of Date.



Out of date assembly drawings can be identified by the exclamation mark icon (⚠).

> Warning: Show Out of Date may take some time depending on how many drawings and parts you have.

> Note: If you switch to another page in Navigator and then go back to the assembly page, the assembly page is refreshed and you will not be able to see which drawings are out of date again until you click Show Out of Date.

Updating an Existing Assembly Drawing

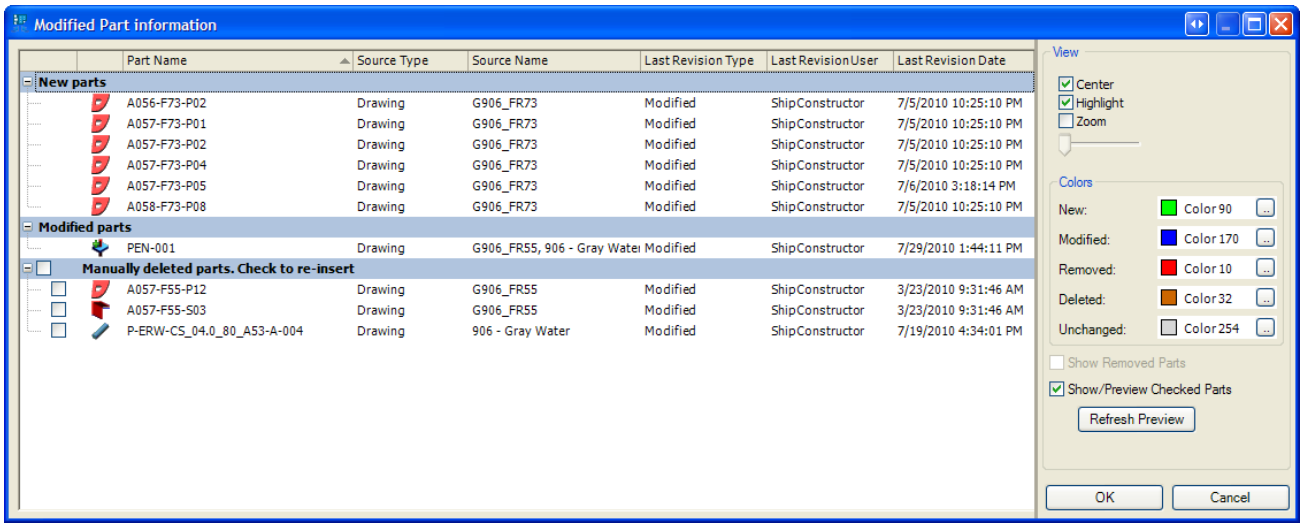
Assembly drawings that are out of date can be updated instead of re-created. This lets you preserve all the customization work done on an assembly drawing, as only the modified geometry of the parts is modified.

Items that are affected during a drawing update include the following:

- All parts will be updated to reflect the latest geometry in the source model drawings.
- The Bill of Materials (BOM).
- All keywords.

To update an assembly drawing

1. Choose SC Assembly > Update Drawing to start updating the drawing.
2. ShipConstructor determines all parts that need to be updated.
3. The Modified Part Information window appears.


Modified Part information
Part Name
Source Type
Source Name
Last Revision Type
Last Revision User
Last Revision Date
New parts
A056-F73-P02 Drawing G906_FR73 Modified ShipConstructor 7/5/2010 10:25:10 PM
A057-F73-P01 Drawing G906_FR73 Modified ShipConstructor 7/5/2010 10:25:10 PM
A057-F73-P02 Drawing G906_FR73 Modified ShipConstructor 7/5/2010 10:25:10 PM
A057-F73-P04 Drawing G906_FR73 Modified ShipConstructor 7/5/2010 10:25:10 PM
A057-F73-P05 Drawing G906_FR73 Modified ShipConstructor 7/6/2010 3:18:14 PM
A058-F73-P08 Drawing G906_FR73 Modified ShipConstructor 7/5/2010 10:25:10 PM
Modified parts
PEN-001 Drawing G906_FR55, 906 - Gray Wate Modified ShipConstructor 7/29/2010 1:44:11 PM
Manually deleted parts. Check to re-insert
A057-F55-P12 Drawing G906_FR55 Modified ShipConstructor 3/23/2010 9:31:46 AM
A057-F55-S03 Drawing G906_FR55 Modified ShipConstructor 3/23/2010 9:31:46 AM
P-ERW-CS_04.0_80_A53-A-004 Drawing 906 - Gray Water Modified ShipConstructor 7/19/2010 4:34:01 PM
View
Center
Highlight
Zoom
Colors
New: Color 90
Modified: Color 170
Removed: Color 10
Deleted: Color 32
Unchanged: Color 254
Show Removed Parts
Show/Preview Checked Parts
Refresh Preview
OK
Cancel

4. You can use the Modified Part Information window to inspect parts that have changed during this update.
5. Click OK to finish the drawing updating.

Insert Keywords into an Assembly Drawing

Usually your assembly drawing template includes the keywords that you want included in assembly drawings. However, there may be certain keywords that you want to manually insert into a few assembly drawings. Once inserted, the keywords will remain as their default text until the assembly drawing is updated.

Inserting keywords into an assembly drawing is similar to inserting keywords into an assembly drawing template, except you use SC Assembly > Insert Keywords (page 491).

To insert keywords into an assembly drawing

1. In the assembly drawing, activate paper space.
2. Choose SC Assembly > Insert Keywords (page 491) to open the Insert Keyword window.
3. Select the keywords to insert.
4. You can change the default text settings now in this window or later using AutoCAD.
5. Click OK.
6. Select the location for each keyword selected.

Note: To change the prefix or postfix text of the keyword, use AutoCAD's property window. When a keyword is updated, the prefix and postfix text will not change.

Update All Keywords

To update keywords

1. Choose SC Assembly > Update Drawing (page 490).
2. The keywords in the drawing update regardless of the status of the parts.

OR

1. Choose SC Assembly > Update All Keywords (page 492).
2. The keywords in the drawing update without updating the parts.

OR

1. Select keywords you wish to update.
2. Choose Update on the right-click menu.

BOM Tables

Insert a New Empty Spool BOM Table

Note: In order to insert a BOM, there needs to be one defined in Manager.

Bills of materials (BOM) in ShipConstructor are highly configurable. For the full functionality overview and for instructions on creating BOM definitions, see Bill of Materials (page 45).

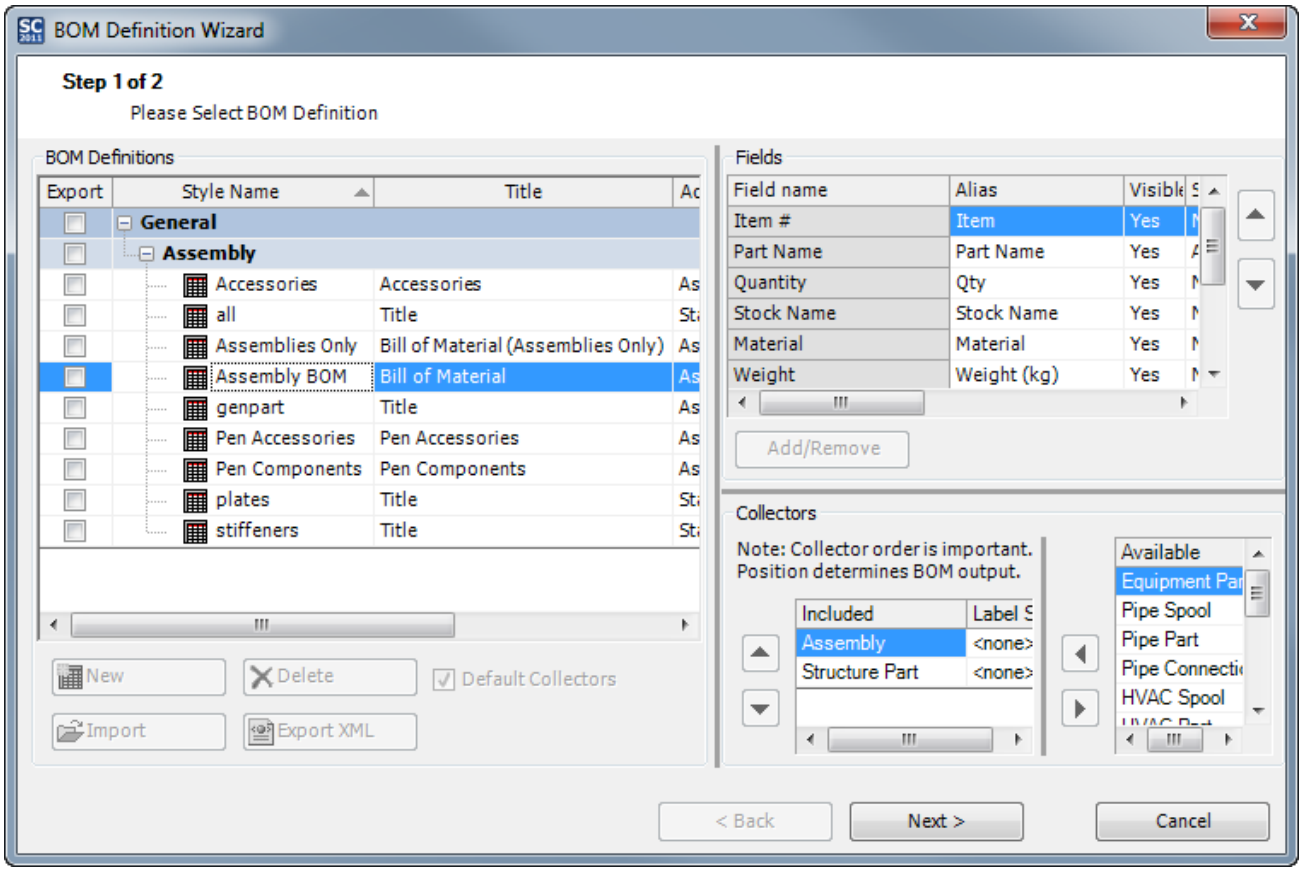
ShipConstructor uses AutoCAD tables to present the BOM data. Tables created using the Insert Empty BOM Table command have specific properties:

- They use the column headings specified by the BOM Definitions window. Column headers are updated every time the tables are updated.
- They are populated with data extracted from the database.
- They use the AutoCAD title, dimension, and table style specified by the BOM Definitions window.
- They are filled after the production drawings are updated and by running the SCUPDATEBOMS command.
- They reference one data master list per BOM definition.
- They reference the same item numbers regardless of layout.

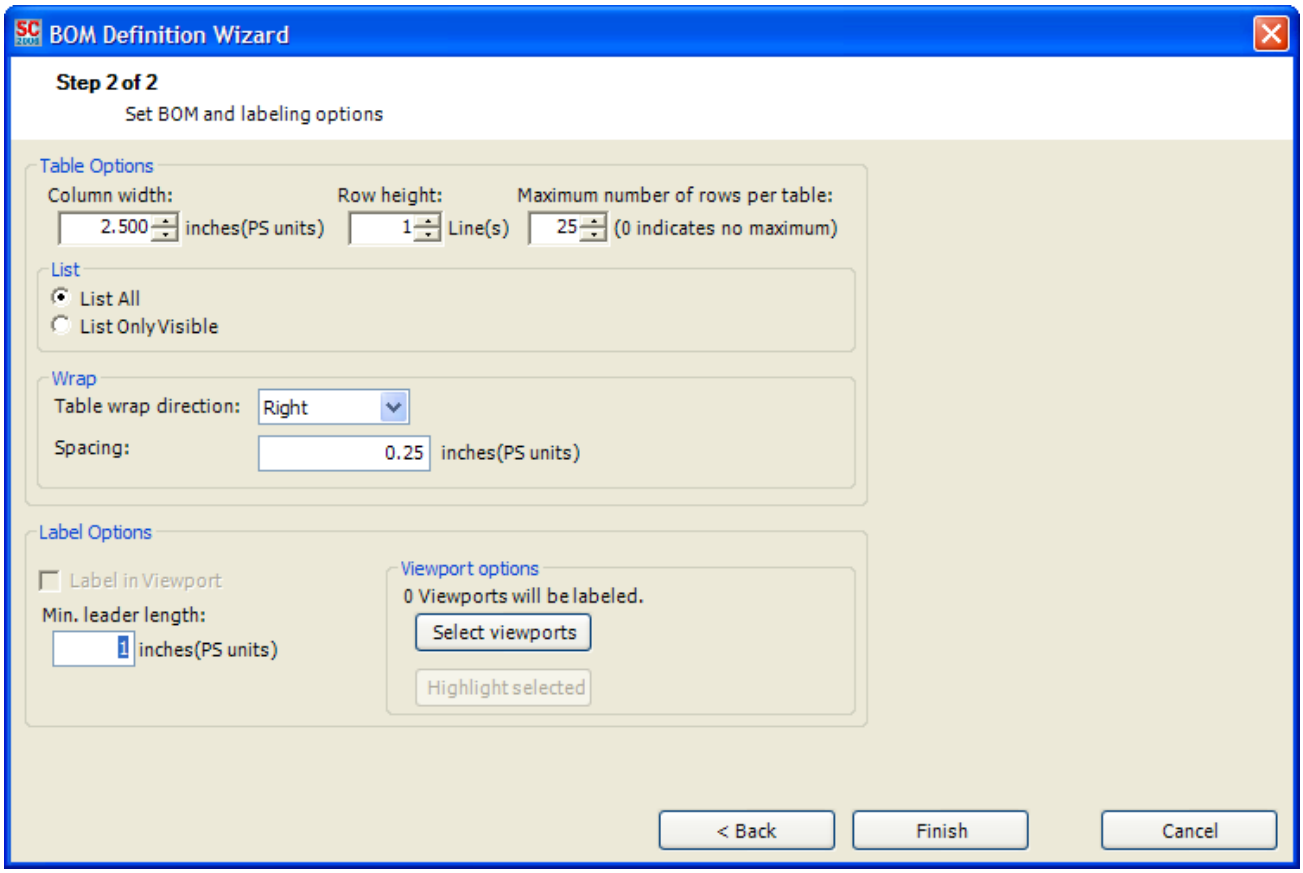
To insert an empty BOM table

Note: You must be in paper space to insert a spool table.

1. Choose SC Assembly > Insert BOM Table. The BOM Definition Wizard appears.



2. Select the appropriate BOM definition and click Next. The settings particular to the BOM definition are displayed for reference after BOM selection.



3. Various options available for BOM and labeling may be specified in step 2. See Insert Empty BOM Wizard (page 105).
4. Click Finish.
5. Depending on the Collectors included in the BOM definition, there may be additional attributes that can be selected. If there are any such attributes, a window appears. The Structure Part Collector that is normally included in a assembly stock BOM contains such options.
6. Accept the default options to ensure that everything in the spool is listed and then click OK.
7. If the drawing is in model space it will automatically switch to paper space to place the BOM.
8. A sample table is created using the AutoCAD table style specified by the BOM Definition. Select the location for the table.
9. You may choose to prearrange more than one table if you anticipate table wrapping (BOMs with more items than specified by the Maximum number of rows property.

Update BOMs

To update all BOM tables

1. Choose SC Assembly > Update BOMs (page 495).
2. All the BOM tables are updated from the objects contained in the drawing.

Edit BOM Collector Options

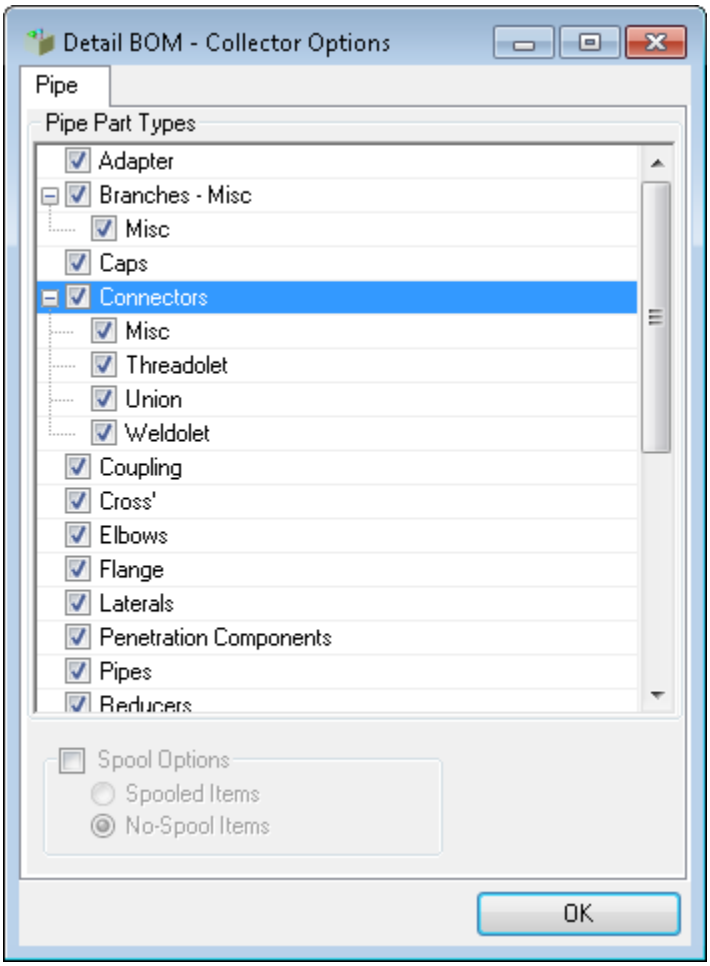
This command allows the user to alter a Bill of Materials Collector options. For example, the Pipe Collector allows the user to select which object types to include/exclude. By using this command you can adjust the filtering of object types to suit your needs.

To change the BOM collector options

1. Choose SC Assembly > Edit Collector Options (page 495)

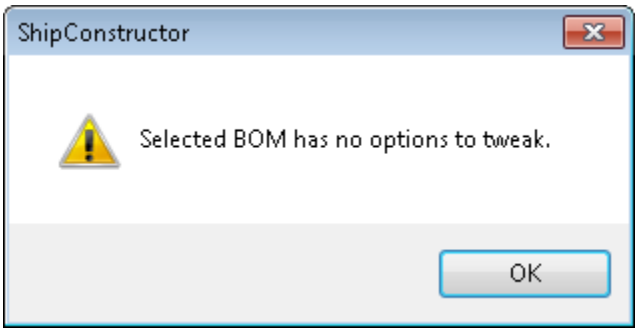
   Select Table:

2. Select the BOM table.



3. The Adjust the Collector options as necessary.

Note: you will be notified if the BOM that you selected has no configurable Collector options.



Viewport Display Options

List only Visible

There is smart functionality in the BOM which will either list only those parts which are visible to the user, or list all parts regardless if they are visible or not.

To toggle the list only visible variable in the BOM

1. Choose Toggle list only visible from the production drawing menu.
2. Information on the command line will inform you whether the variable is set or not.

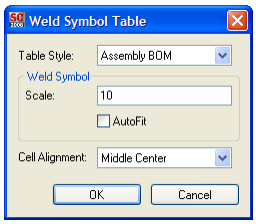
Note: You will want to run Update BOMs to see changes from this command in the BOM table. If all parts are visible, there will be no difference.

Insert a Weld Symbol Table into an Assembly Drawing

Weld Symbol tables are normally inserted into the assembly drawing templates. However, if you want to insert a weld symbol table for a certain drawing, the insertion method remains the same as for inserting it into an assembly drawing template. The drawing's weld symbol tables will need to be updated after inserting in the empty one.

To insert a Weld Symbol Table

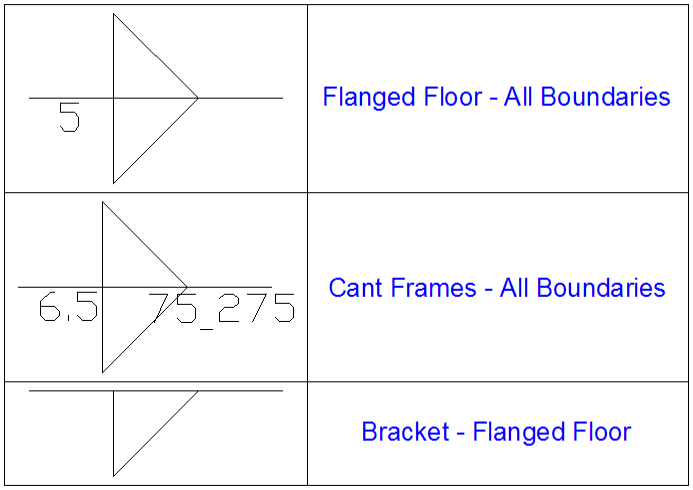
1. In the assembly drawing template, activate paper space.
2. Choose SC Assembly > Insert Weld Symbol Table



3. Select a table style, scale for the Weld Symbols, and an aligment for the table's cells.
4. Click OK.
5. Select a location for the empty table.

Update Weld Symbol Tables in an Assembly Drawing

Instead of having to run a full update on the entire drawing, you can update only the weld symbol tables.

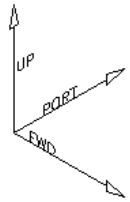


To update all Weld Symbol Tables

1. Choose SC Assembly > Update Weld Symbol Tables

Insert an Orientation Icon into an Assembly Drawing

Orientation icons are useful in displaying the directions when it is not clear how the assembly is oriented.

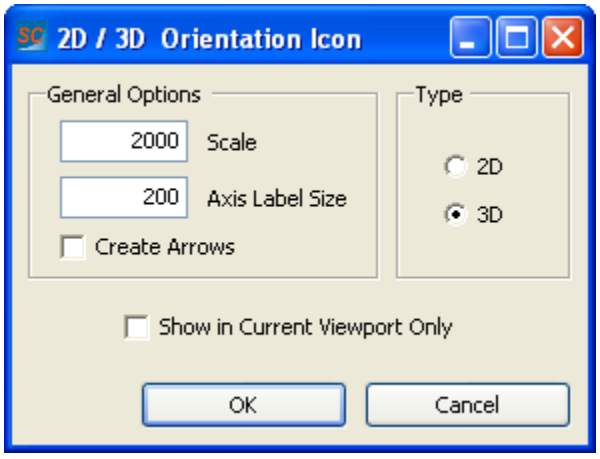


3D Orientation icon

To insert an orientation icon into an assembly drawing

1. Choose SC Assembly > Orientation Icon (page 490)

   The 2D/3D Orientation Icon window appears.



2. Select the type of icon (3D or 2D), the length of the direction lines (Scale), and any other options.
3. Click OK.
4. Select the location for the icon.

Mark an Assembly's CG Position in an Assembly Drawing

You can insert a mark into an assembly drawing indicating the assembly's center of gravity (CG). This can be important for lifting documentation and when you are planning to move heavy assemblies. The CG position is located using a SConCGPoint object. This object cannot be moved and will be updated if the drawing is updated.

If you need to rotate the assembly in the assembly drawing (see Rotate an Assembly in an Assembly Drawing (page 237)), then you first need to mark the assembly's CG position and explode the SConCGPoint.

To mark an assembly's CG position

1. In the assembly drawing, make sure you are in model space.
2. Choose SC Assembly > CG Point (page 491)

   ShipConstructor inserts a SConCGPoint object at the CG point.

Change the View in an Assembly Drawing

When you change the view of an assembly, all annotations will not move. This is because annotations are always in paper space.

To change the view in an assembly drawing

1. Double-click on a view to activate model space.
2. Use the view controls to change the view.

Rotate an Assembly in an Assembly Drawing

You may want to rotate the assembly within an assembly drawing to view the assembly from a different angle. For example, you may want to rotate an assembly so it appears upside down if that is the way it will be assembled. The preferred procedure is to orbit around to the correct view position instead of rotating parts.

When you create assembly drawings, only visible sub-assemblies are annotated. If you later rotate the assembly, you may expose otherwise hidden (and un-annotated) sub-assemblies. After rotating an assembly, you must re-auto-annotate the drawing to ensure that all components are labeled.

Note: The SConCGPoint object will not be rotated when the parts are rotated. You can move the CG point by exploding it. However, updating the drawing will not update the exploded CG point.

To rotate an assembly in an assembly drawing

Note: Before rotating an assembly, make sure you mark its CG position (see Mark an Assembly's CG Position in an Assembly Drawing (page 236).)

1. In the assembly drawing, double-click within the viewport to switch to model space.
2. Choose Modify > 3D Operation > Rotate 3D.
3. Type all and press Enter to select all objects.
4. Press Enter to finish selecting objects.
5. Type the axis to rotate the assembly about and press Enter.
6. Press Enter to accept the default rotation point (0, 0, 0).
7. Type the angle to rotate the assembly and press Enter.

Insert a Quality Control Matrix into an Assembly Drawing

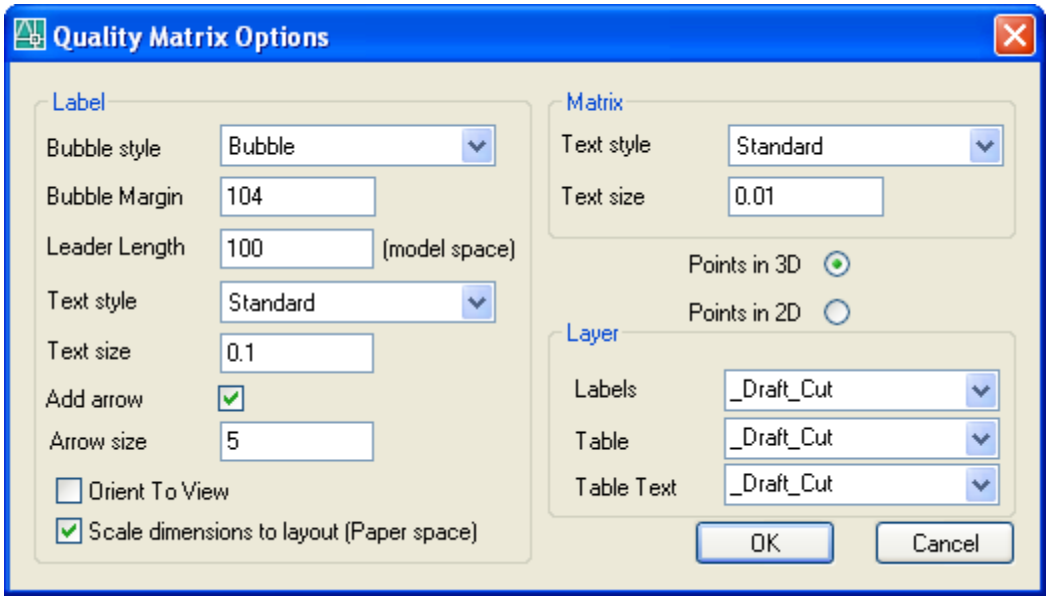
A quality control matrix is a table of dimensions that you can insert into an assembly drawing and that production can use to verify the accuracy of an assembly after its fabrication.

After fabrication, production can measure the distances between points on the physical assembly and enter the values into the empty fields of the quality control matrix.

Note: You can insert a quality control matrix into any ShipConstructor drawing, but they are most useful for assembly drawings.

To insert a quality control matrix into an assembly drawing

1. In the assembly drawing, double-click within the main viewport to switch to model space.
2. Choose SC Utilities > Create Quality Matrix (page 462) to open the Quality Matrix Options window.



3. Set the options and click OK.
4. Click the points on the assembly that you want to include dimensions between. For example, click points on one plate part to generate a quality control matrix for that panel.

Manual Labeling from BOM

You can label an assembly drawing either automatically or manually. Manual Labeling allows you to individually place labels where you want them without having to go though a move process that you would have to do from the automatically generated labels.

To manually label a part

1. Choose SC Assembly > Label > Manual Label from BOM (page 496)
2. If there are multiple BOM tables in current layout, you will be prompted to select a BOM table for labeling. If there is only one BOM in the layout it will choose that. The first column text in the BOM table will be used as the label text and label style of the BOM or in the corresponding collectors of the BOM will be used if specified.

To find what label style it is using go to Manager > General > Production Output > Bill of Materials. Select the BOM definition used to create the table. If the collector has a label style then it will be that style. If it is <none> then the label style will be the BOMs label style.

```
Select a part to label:
```

3. Select a part to label. The selected point will be the position where the arrow points to.

```
Specify label position (1 seg)[Multi-segment]:
```

4. Specify label position. (1 seg) indicates that the label style has the Num Segments value of 1. If your label style has more than one segment then you will be prompted to pick the remaining segment points. If you type in an ‘M’, you are permitted to select more leader points than the style has. Press Enter will finish the Multi-segment label mode.

```
Select a part to label [Copy]:
```

5. After you create the first label, you can switch to Copy Label mode by typing in ‘C’. In this mode, you can continue to label parts with only having to select the part and not the label position. The label geometry will be the same as the previous label.

```
Select part to copy label [lAbel]:
```

6. You can switch back to manual label mode by typing ‘A’.
7. Press Esc at anytime will end the labeling.

To copy a label for labeling another part

1. Choose SC Assembly > Label > Copy Label from BOM (page 496)

```
Select the source label:
```

2. Select the source label. The system will find out the related BOM table for copying label text and styles.

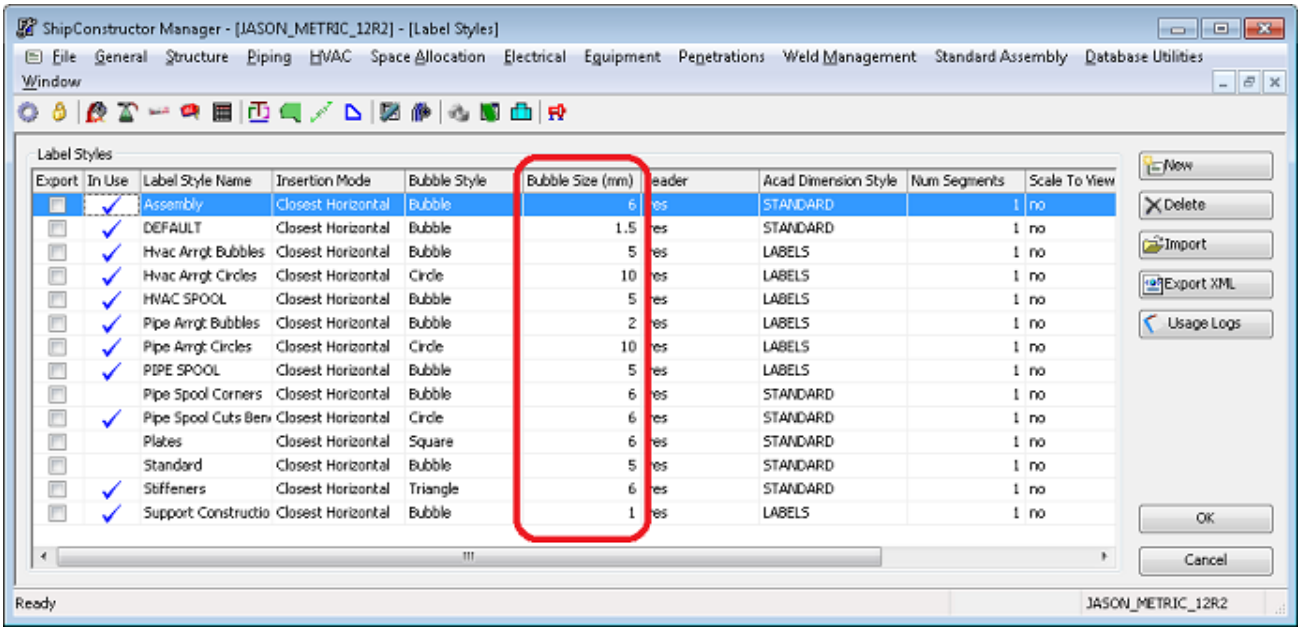
```
Select a part:
```

3. Select a part to label. The selected point will be the position where the arrow points to and all other geometry information will be copied from the source label.
4. Press Esc at anytime will end the labeling.

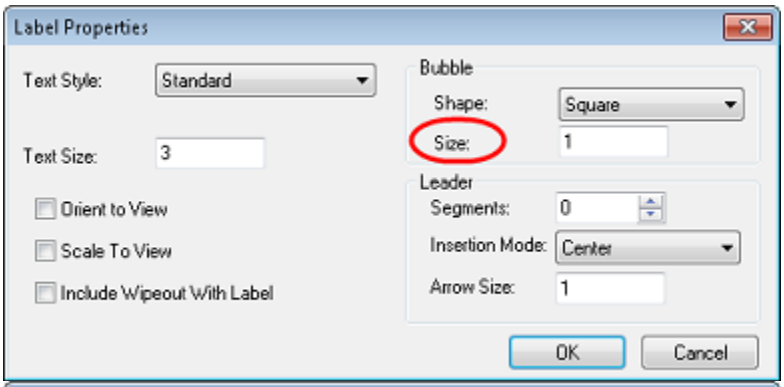
Setting up Label Sizes

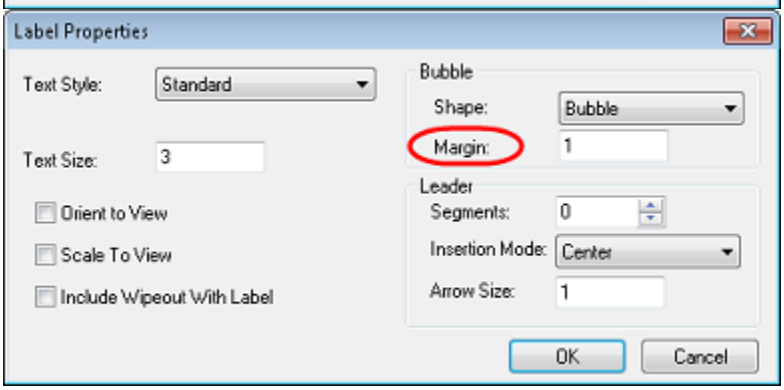
Understanding bubble sizes

In Label Styles catalog in Manager, a parameter that affects the size of label bubbles is called Bubble Size.


ShipConstructor Manager - [JASON_METRIC_12R2] - [Label Styles]
File General Structure Piping HVAC Space Allocation Electrical Equipment Penetrations Weld Management Standard Assembly Database Utilities
Window
Label Styles
Export In Use Label Style Name Insertion Mode Bubble Style Bubble Size (mm) eader Acad Dimension Style Num Segments Scale To View
Assembly Closest Horizontal Bubble 6 es STANDARD 1 no
DEFAULT Closest Horizontal Bubble 1.5 es STANDARD 1 no
Hvac Arrgt Bubbles Closest Horizontal Bubble 5 es LABELS 1 no
Hvac Arrgt Circles Closest Horizontal Circle 10 es LABELS 1 no
HVAC SPOOL Closest Horizontal Bubble 5 es LABELS 1 no
Pipe Arrgt Bubbles Closest Horizontal Bubble 2 es LABELS 1 no
Pipe Arrgt Circles Closest Horizontal Circle 10 es LABELS 1 no
PIPE SPOOL Closest Horizontal Bubble 5 es LABELS 1 no
Pipe Spool Corners Closest Horizontal Bubble 6 es STANDARD 1 no
Pipe Spool Cuts Ben Closest Horizontal Circle 6 es STANDARD 1 no
Plates Closest Horizontal Square 6 es STANDARD 1 no
Standard Closest Horizontal Bubble 5 es STANDARD 1 no
Stiffeners Closest Horizontal Triangle 6 es STANDARD 1 no
Support Constructio Closest Horizontal Bubble 1 es LABELS 1 no
New
Delete
Import
Export XML
Usage Logs
OK
Cancel
Ready
JASON_METRIC_12R2

In the Label Properties window in production drawings, the same setting is called Size or Margin depending on the style selected as shown below.

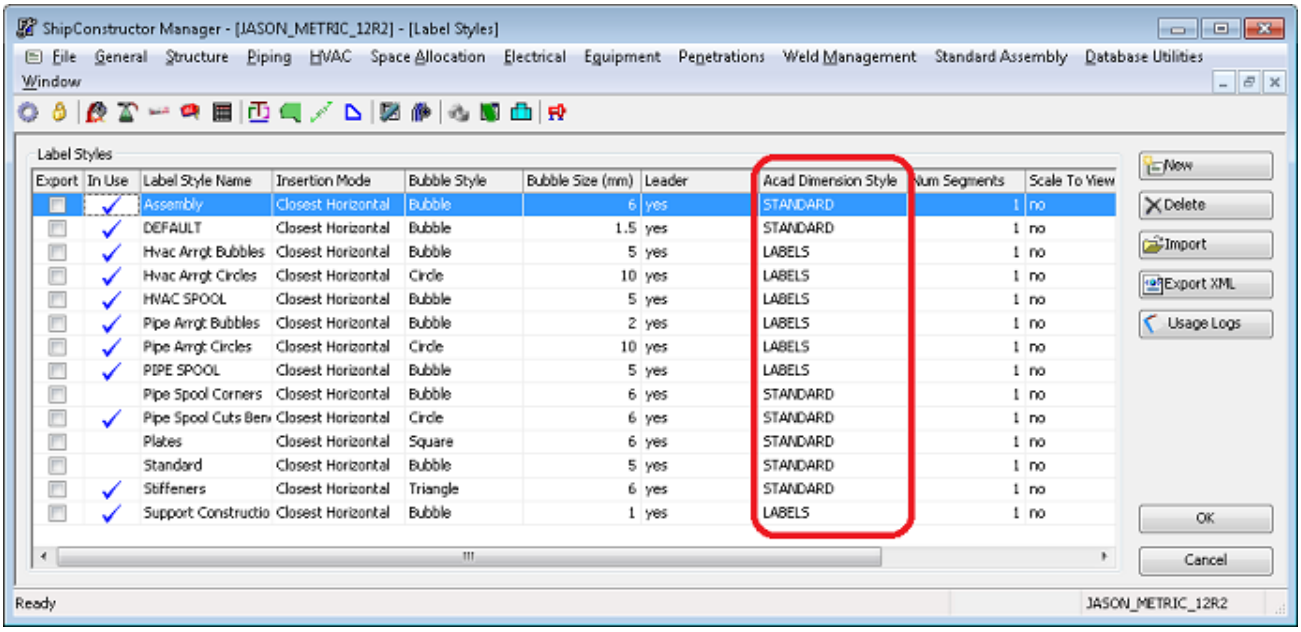




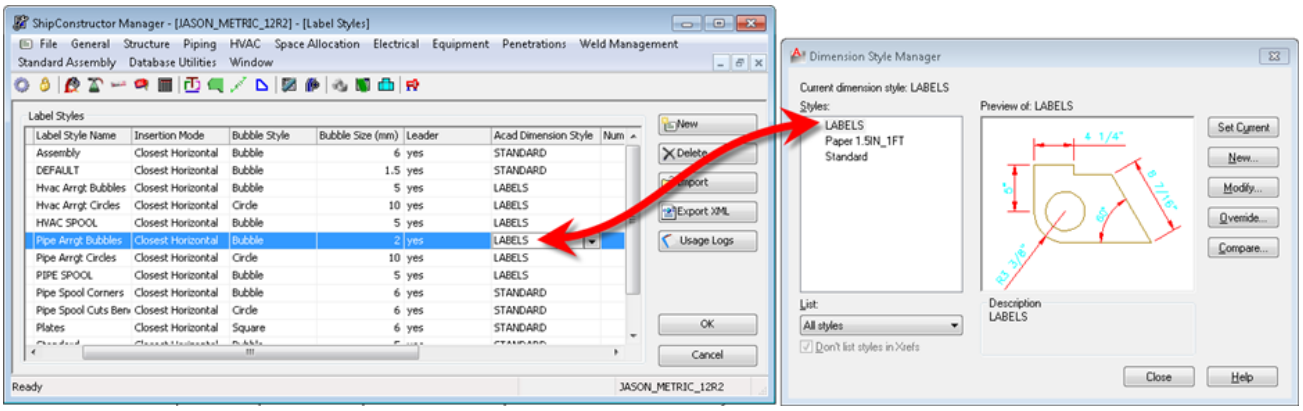
Bubble Style	Visible Size
Circle	Bubble Size (or Margin) represents the radius of the circle
Square	Bubble Size (or Margin) represents the length of an edge of the square
Triangle	Bubble Size (or Margin) represents the length of an edge of the triangle
Octagon	Bubble Size (or Margin) represents the distance between parallel edges
Rectangle	Bubble Size (or Margin) represents the distance between the text and the rectangle. Overall height is Text Height + 2*Bubble Size.
Bubble	Overall height is Text Height + 1.5*Bubble Size.

Setting up Label Text Styles

In Label Styles catalog in Manager, each label style is referenced to a specific dimension style name.

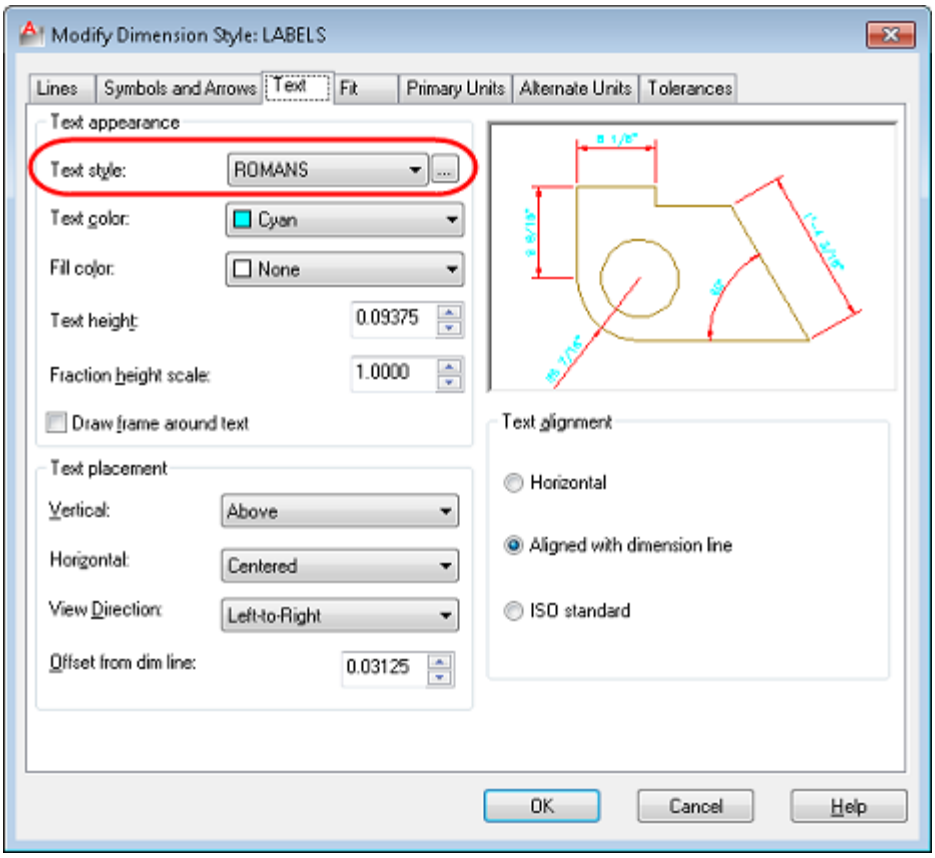

ShipConstructor Manager - [JASON_METRIC_12R2] - [Label Styles]
File General Structure Piping HVAC Space Allocation Electrical Equipment Penetrations Weld Management Standard Assembly Database Utilities Window
Label Styles
Export In Use Label Style Name Insertion Mode Bubble Style Bubble Size (mm) Leader Acad Dimension Style Num Segments Scale To View
Assembly Closest Horizontal Bubble 6 yes STANDARD 1 no
DEFAULT Closest Horizontal Bubble 1.5 yes STANDARD 1 no
Hvac Arrgt Bubbles Closest Horizontal Bubble 5 yes LABELS 1 no
Hvac Arrgt Circles Closest Horizontal Circle 10 yes LABELS 1 no
HVAC SPOOL Closest Horizontal Bubble 5 yes LABELS 1 no
Pipe Arrgt Bubbles Closest Horizontal Bubble 2 yes LABELS 1 no
Pipe Arrgt Circles Closest Horizontal Circle 10 yes LABELS 1 no
PIPE SPOOL Closest Horizontal Bubble 5 yes LABELS 1 no
Pipe Spool Corners Closest Horizontal Bubble 6 yes STANDARD 1 no
Pipe Spool Cuts Ben Closest Horizontal Circle 6 yes STANDARD 1 no
Plates Closest Horizontal Square 6 yes STANDARD 1 no
Standard Closest Horizontal Bubble 5 yes STANDARD 1 no
Stiffeners Closest Horizontal Triangle 6 yes STANDARD 1 no
Support Constructio Closest Horizontal Bubble 1 yes LABELS 1 no
New
Delete
Import
Export XML
Usage Logs
OK
Cancel
Ready
JASON_METRIC_12R2

The actual dimension style should exist in the production drawing. If it doesn't, the Standard dimension style will be used. The dimension style in the production drawing must be named exactly the same as the field in the Label Styles catalog in Manager.

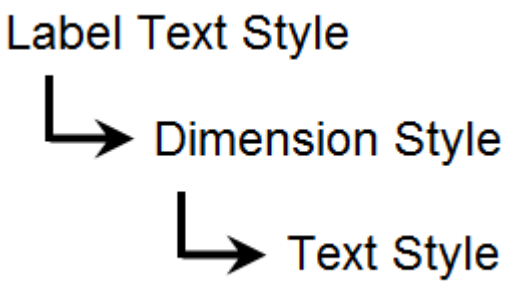


The simplest way to ensure that all production drawings have properly named dimension styles is to predefine required dimension style in production drawing templates.

In AutoCAD, each dimension style is referenced to a Text Style.

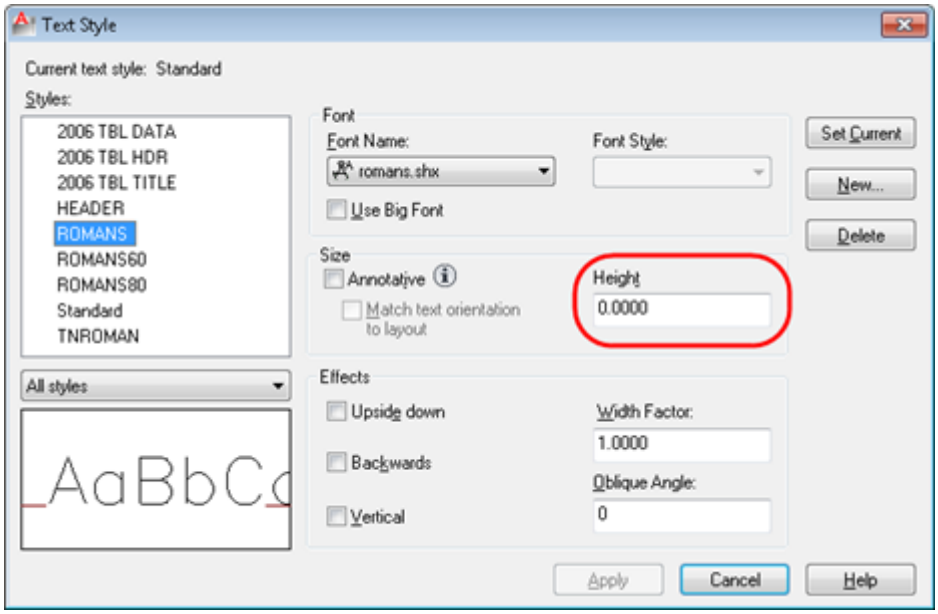


ShipConstructor labels get their text styles from the text style that is assigned to the dimension style that is associated with the label style:



Setting up Label Text Sizes

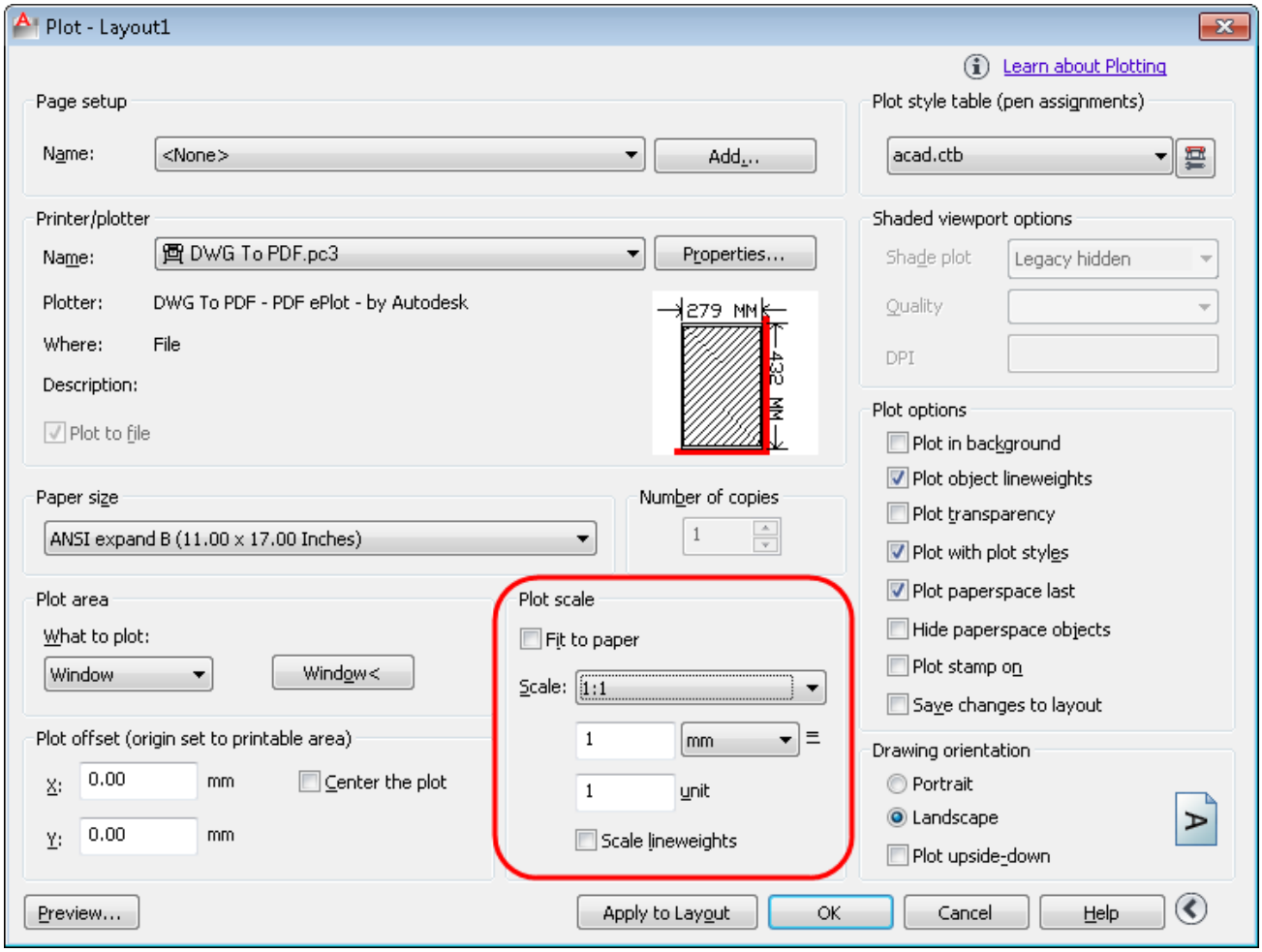
A situation with the text size is similar, but a little bit more complex. If the destination text style has zero height, the height of the label text comes from the associated dimension style. Otherwise, it will be the height from the text style definition. The same convention AutoCAD dimension text uses.



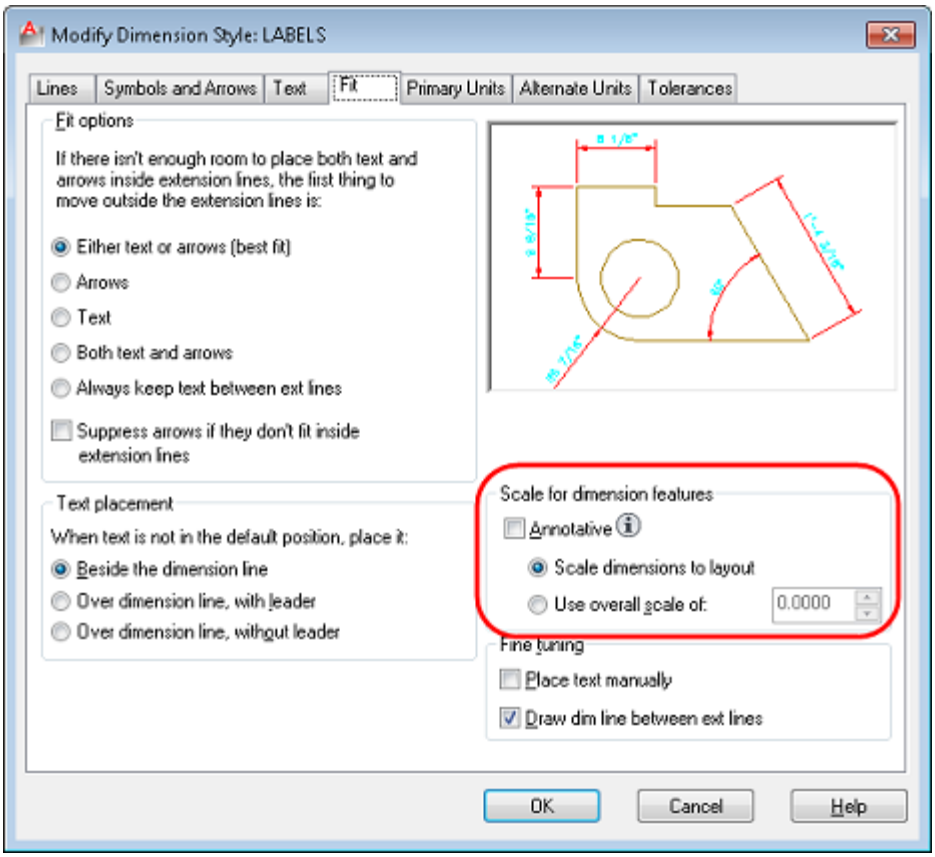
When everything is set, but label bubbles and text still have wrong sizes

When everything is set, but label bubbles and text still have incorrect sizes, the user should check scaling parameters in the production drawing. There are a few scaling parameters to be aware about:

- The Scale to View setting should always be turned off for labels that live in paper space. Scale to View is an old ShipConstructor setting that was introduced back in the times when ShipConstructor labels were placed in the model space. Currently, the only type of the drawing where labels appear in the model space is the plate nest drawing. In the rest of the drawings, labels live in the paper space, so additional scaling to view is not required in these drawings. If the Scale to View setting happens to be turned on for paper space labels, the size of the label will change depending on the current zoom of the layout, which may not be a desired behavior.
- AutoCAD Plot Scale that can be set in the Page Setup dialog affects the overall scale of the entire layout view.



- Dimension styles have their own scaling that affect the text size.



Label sizes are affected by scaling settings of the associated dimension style.

Plot Assembly Drawings

Plot an Assembly Drawing

To plot an assembly drawing

1. In the assembly drawing, make sure you are in paper space. (If you are in model space, only the active viewport will be plotted.)
2. At the command line type MVIEW and Enter.

   ```
   [ON/OFF/Fit/Shadeplot/Lock/Object/Polygonal/Restore/2/3/4] <Fit>:
   ```

3. Enter Shadeplot and press Enter.

   ```
   Shade plot? [As displayed/Wireframe/Hidden/Rendered] <As displayed>
   ```

4. Enter Hidden and press Enter.

   ```
   Select Objects:
   ```

5. Pick the viewport or viewports for which you want to hide lines.
6. PLOT the drawing.

Inspect a Plotted Assembly Drawing

After plotting an assembly drawing, visually inspect the plotted drawing for format and content. Make sure things like 3D orientation icons and weld symbols are included and that every part is annotated.

Labeling

Relabel All

This command re-labels all the parts from the BOM across all sheets and places the labels on the _ANNOTATION layer.

Labels are generated only once. Subsequent commands only update the label text to reflect the Bill of Materials.

Even though labels are placed in Paper Space, they now track the objects in Model Space so that if the user pans, zooms, or orbits in the viewport, the labels track accordingly and remain synchronized to the objects that they represent.

To re-label parts from BOMs

1. Choose SC Assembly > Label > Relabel All (page 496).

Manual Label from BOM

Label parts from a BOM. This is an alternative method to automatic labeling. The label style and text contents are used from the selected BOM. The selection of the leader arrow is the pickpoint where you selected the part. The number of leader segments follows the label style setting.

This command can be used in conjunction with Copy Label from BOM so complete the labeling. If you select a part that has been previously labeled then the previous label is removed.

To manual label from BOM

1. Choose SC Assembly > Label > Manual Label from BOM (page 496)

Copy Label from BOM

Lets you create labels that are copies of an existing label with the contents changed to reflect the first BOM column of the selected part. The copied label will be created with the label text and styles from the BOM table and geometry information from the source label.

To manual label from BOM

1. Choose SC Assembly > Label > Copy Label from BOM (page 496)

Relabel from BOM

This command only labels items from select tables and viewports. Labels are placed on the _ANNOTATION layer.

Labels are generated only once. Subsequent commands only update the label text to reflect the Bill of Materials.

Even though labels are placed in Paper Space, they now track the objects in Model Space so that if the user pans, zooms, or orbits in the viewport, the labels track accordingly and remain synchronized to the objects that they represent.

To re-label parts from BOMs

Choose SC Assembly > Label > Relabel from BOM (page 497).

1. Select the BOM for which to generate labels from.

Relabel from Parts

This command will label all the parts selected by the user from the selected table.

To re-label parts from Parts

Choose SC Assembly > Label > Relabel from Parts (page 497).

1. Select the BOM for which to generate labels from.

Label to part edges

This feature allows user to place labels on borders of parts instead of piece marks. Since there is only one piece mark on each part, it's easy to find the location and lay the label arrow on it. It's much more complicated to label on borders, because there could be none or many locations to be labelled on. Sometimes, it is hard to say where the best place is to put the label on. Several rules were setup in the searching algorithm trying to get the best point on borders.

For detail explaination with diagrams, please refer to the Structure help file for Assembly drawings.

Terminology

Bend Angle

By connecting point 1, 2 and 3, we can get 2 straight line segments, and A indicates how much the second line direction turned away from the first.

Short edge, Long edge, Label Edge Length Factor

Label Edge Length Factor is a percentage calculated by dividing the length of the testing edge by the length of the longest edge in the part. If the percentage is larger than the set value, the edge is a Long edge, otherwise a Short edge. Short edges will never get labelled on.

Distance to other parts

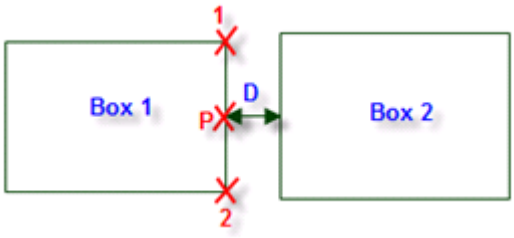
The sum of the distances between a point and all other parts' extents.

Convex Hull points

Those red points in the diagram are Convex Hull points. Convex Hull may be easily visualized by imagining an elastic band stretched open to encompass the given object. Some real edges are overlapped with the convex hull lines, and we will pick one of the best edges from them to lay the label on.

Convex Hull algorithm get a time complexity of O(n log n). It is an efficient way to find out most of the corner points.

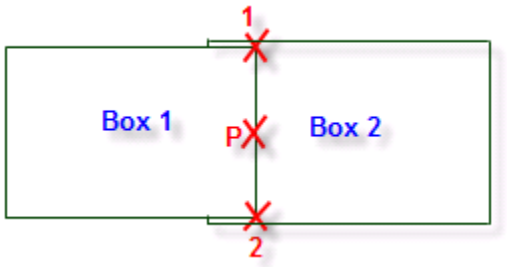
OverlapOnBorder



The midpoint of the edge is very close or overlaps to a visible edge from other parts.

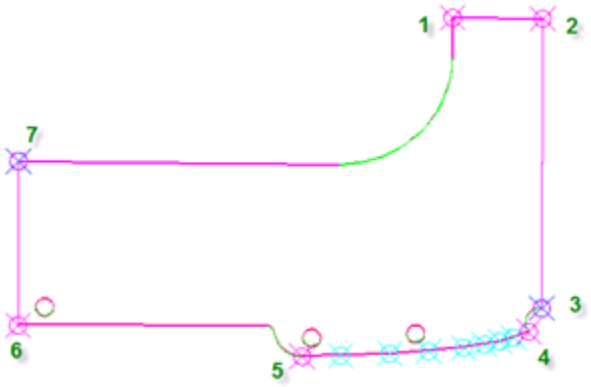
The distance (D) between the midpoint (p) and the closest edge on Box2 is less than SCLABELADJACENTTOL.

OverlapOnPart



The midpoint of the edge is inside of another part.

Find Group Edges



After HLR and 3D-2D transforms, we might get tens or hundreds of curves and lines. We need a fast algorithm to figure which edges are on the border and where is the midpoint on each edge.

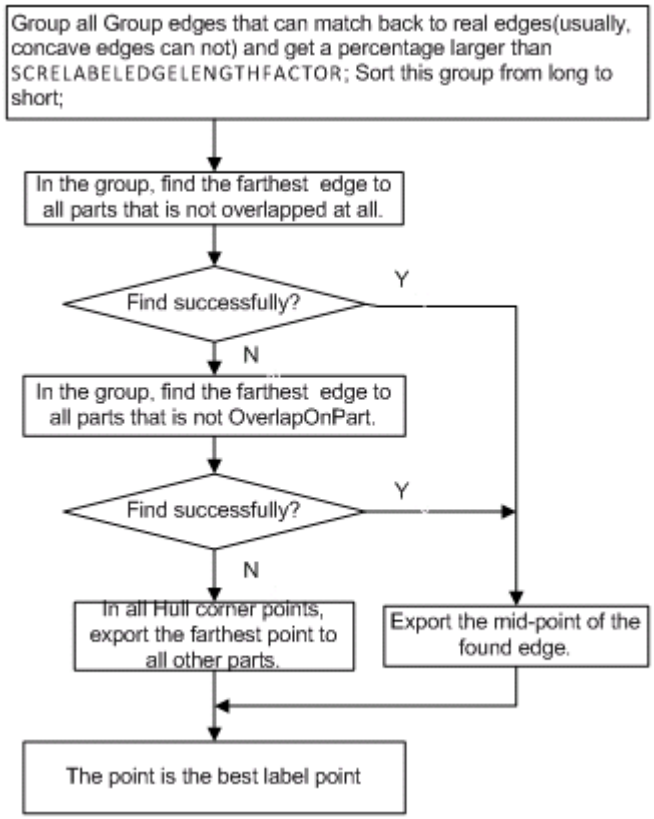
To get a good performance, we are using Convex Hull algorithm to find those convex points on the border; then matching them back to curves on the part to figure out the original lines and midpoints. Because we only got convex hull points, those curves in concaved border edges will not be found out and used. For example, in the diagram above, those purple and blue points are those convex hull points returned from the algorithm. Because we want to label to a midpoint of a edge, we go over those points one by one and mark any point that has a Bend Angle bigger than system setting as group edge point. Points 1 to 7 are final group points we got. Group Edge (1,2), (2,3), (3,4), (4,5), (6,7) are the 5 possible label edges. Because we matched those real 3d edges on the part with those group edges, user can orbit the 3D model and those labels can keep point on it.

Rules to find best label points

After Group Edges on each part were located from a viewport, the following rules are used to find the best label point on a part. We always want to put labels on a Long edge, but since other rules need to be considered, we first find those Long edges (longer than Label Length Factor) and test the following rules in this group.

1. The mid point of an edge that has the largest Distance to Other Parts in the longest NoneOverlap group edges.
2. The mid point of an edge that has the largest Distance to Other Parts in the longest not OverlapOnBorder group edges.
3. The Hull corner point that has the largest Distance to Other Parts.

Rules chart


Group all Group edges that can match back to real edges(usually, concave edges can not) and get a percentage larger than SCRELABELEDGELENGTHFACTOR; Sort this group from long to short;
In the group, find the farthest edge to all parts that is not overlapped at all.
Find successfully?
Y
N
In the group, find the farthest edge to all parts that is not OverlapOnPart.
Find successfully?
Y
N
In all Hull corner points, export the farthest point to all other parts.
Export the mid-point of the found edge.
The point is the best label point

Label on Visible Edge On/Off

If the value is true, all labels will be laid on HLR points (part centres), otherwise they will be on edges.

To run the command

1. Choose SC Assembly > Label > Label on Visible Edge On/Off (page 497)
2. Set the value on command line

Adjacent Part Edge Tolerance

If the gap between a midpoint of a group edge and any other visible edge from any other objects is smaller than SCADVLABELADJACENTTOL, the midpoint will not be used as a label point.

If a label arrow points to an edge of an object that has other objects close to it, it is hard to tell which object the arrow is pointing to. This setting is trying to filter out those confusing cases.

To run the command

1. Choose SC Assembly > Label > Label on Visible Edge On/Off (page 497)
2. Set the value on command line

Curved Plates in Visible Edge Detection On/Off

If it is false, the frame of the curved plate will be used in HLR projection instead of the plate solid. As you can see, it is not very realistic since the frame won't hide anything behind it. The advantage of this is that you will get a much faster results because otherwise the surface will be used in HLR, which might bring in hundreds of thousands more points into the calculation.

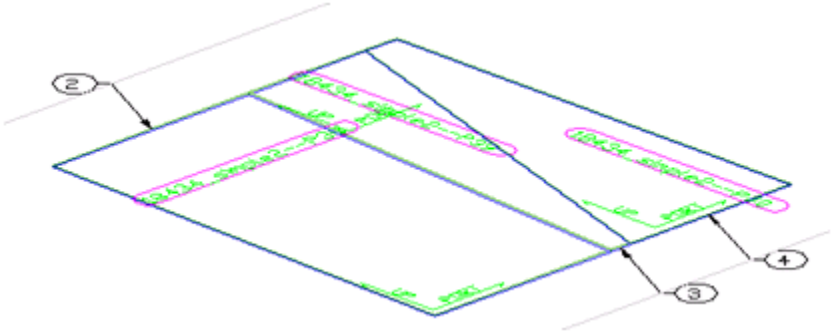
To run the command

1. Choose SC Assembly > Label > Curved Plates in Visible Edge Detection On/Off (page 497)
2. Set the value on command line

Edge Length Filter

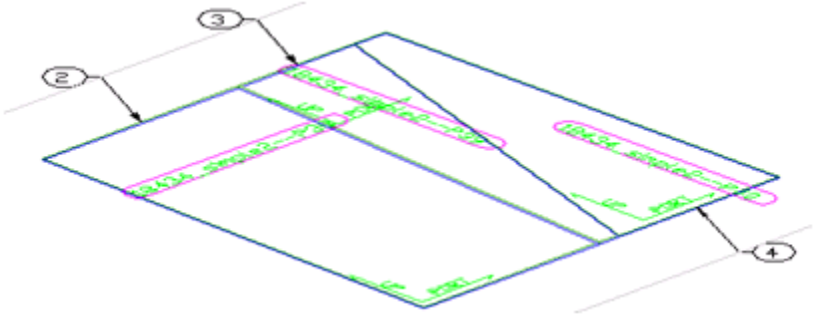
It is a percentage value calculated by dividing the length of the testing edge with the length of the longest edge in the part. The testing edge will not be picked as a qualified labeling edge if its percentage is smaller than this setting.

We want to find out all long edges on a object first, then try to find the best long edge that are far from other objects to label. These setting might bring some confusion like the following sample.



Length Filter <2.000000>

Label 3 was picked from a short edge because the setting is too small.



Length Filter <25.000000>

Label 3 is on a longer edge.

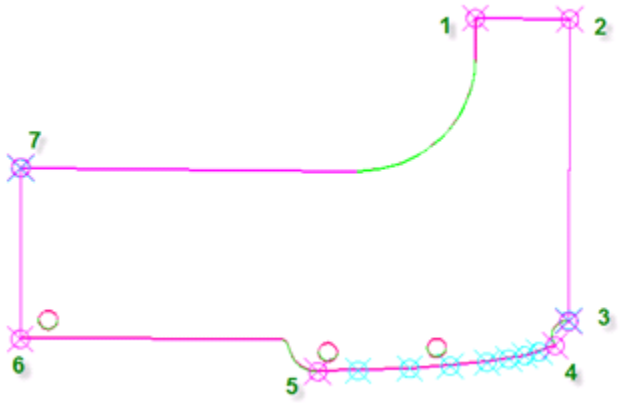
To run the command

1. Choose SC Assembly > Label > Edge Length Filter (page 498)
2. Set the value on command line

Edge Determination Minimum Angle

When Hull points were caught from HLR, we'll go through point to point with straight lines. If an angle between any 2 adjacent lines is bigger than this setting, the intersect point will be counted as a group line end point.

Group edge is used as the base element for a label to point. The length and the position of a group edge will be used to decide whether it can be used as a label edge.



Like in this picture, points between 4 and 5 formed a group edge because no big angle elbow between them.

To run the command

1. Choose SC Assembly > Label > Edge Determination Minimum Angle (page 498)
2. Set the value on command line

Label Reset Automatic Settings

All label settings come originally from Project Settings in Manager. Once user changed a setting through a label command, the current value will be saved and used from current drawing. If you run this reset command and confirmed with a 'yes', all current label settings will be replaced back to the Project Settings again.

To run the command

1. Choose SC Assembly > Label > Label Reset Automatic Settings (page 498)
2. Confirm whether you want to reset on command line

BOM Revisions

When BOM tables are updated, they are rebuilt each time to reflect the current state of the drawing. This does not allow the user to see how the drawing changes affected the BOM tables. It also may renumber the items, causing existing labels to be incorrect.

BOM Revisions allow the user to save the current state of the Bills of Material in the drawing (or layout), so that when the BOM tables are updated, they will be able to compare the new BOM data with the old BOM data and display those changes to the user.

To prevent the original BOM tables appearance from changing, new items in the BOM are appended to the end of the table, and removed items have their text changed to use strikeout. Any changes to existing items are simply updated into their rows. This allows the items to maintain their original item numbering, preventing the user from having to relabel everything.

To use BOM revisions best, a Revision column should be added to the BOM definitions. This column will display one of three possible values:

- nothing, for when no revisions exist
- the current revision, such as "Rev.0"

- the “modified” state, which is the current revision prepended with “*”, such as “*Rev.0”. This state indicated that something in this row was modified during an update.

BOM Revisions can be defined for any paperspace layout in production drawings. Alternatively, the user can define a revision for the entire drawing (all layouts), instead of for just the current layout.

An example of how the BOM table evolves is shown:

Item #	Part Name	Revision
1	PEN-008-Collar	
2	E-90LR-CS_02.0_ST-A234-001	
3	E-90LR-CS_02.0_ST-A234-002	
4	E-90LR-CS_02.0_ST-A234-005	
5	P-ERW-CS_02.0_80_A53-A-019	
6	P-ERW-CS_14.0_80_A53-A-007	

This is a BOM table in a Pipe Arrangement drawing after creation. The user defines a revision called “A”. The empty Revision column is filled with this name.

Item #	Part Name	Revision
1	PEN-008-Collar	A
2	E-90LR-CS_02.0_ST-A234-001	A
3	E-90LR-CS_02.0_ST-A234-002	A
4	E-90LR-CS_02.0_ST-A234-005	A
5	P-ERW-CS_02.0_80_A53-A-019	A
6	P-ERW-CS_14.0_80_A53-A-007	A

Then they update the drawing. Some parts have been added, and others have been removed. The result looks like this:

Item #	Part Name	Revision
~~1~~	~~PEN-008-Collar~~	*A
2	E-90LR-CS_02.0_ST-A234-001	A
3	E-90LR-CS_02.0_ST-A234-002	A
4	E-90LR-CS_02.0_ST-A234-005	A
5	P-ERW-CS_02.0_80_A53-A-019	A
~~6~~	~~P-ERW-CS_14.0_80_A53-A-007~~	*A
7	P-ERW-CS_02.0_80_A53-A-010	*A
8	P-ERW-CS_02.0_80_A53-A-017	*A

The changed parts have the “*A” identifier. The removed parts used strikeout text. New rows are appended to the end, continuing the item numbering, and not re-ordering it.

New Revision

Revisions can be created at any time: the user simply needs to provide a name, and an optional description. The new revision name must not be used in the layout (or drawing, if creating a revision for all layouts). The revision saves the current user and the date and time of creation.

If this is the first revision in the drawing, the empty Revision column for all BOM tables in the layout (or drawing) are changed to the revision name. If it's not the first revision, then all the rows with in the "*" identifier are changed to the new revision. This way, only items that have changed during the BOM updating are tagged as belonging to the new revision.

To create a revision

1. Choose SC Assembly > BOM Revisions > New BOM Revision in Current Layout (page 499) or SC Assembly > BOM Revisions > New BOM Revision in All Layouts (page 499)
2. Provide a name for the revision

   ```
   Enter a name for the new revision:
   ```

3. Provide a description for the revision (this can be left blank)

   ```
   Enter a description for the new revision <none>:
   ```

4. The revision is created in the specified layouts

   ```
   Revision 'Rev.A' created in layout 'Master BOM'
   ```

   Or

   ```
   Revision 'Rev.A' created in 2 layouts
   ```

Delete Revision

When deleting revisions, only the current revision can be deleted. Any BOM item with the current revision is changed to be the previous revision with the "*" identifier. If no previous revision exists, then the Revision column is left blank.

Using the "all layouts" command to delete revision requires that the current revision in all those layouts have the same name.

To delete a revision

1. Choose SC Assembly > BOM Revisions > Delete BOM Revision in Current Layout (page 499) or SC Assembly > BOM Revisions > Delete BOM Revision in All Layouts (page 499)
2. The current revision is deleted

   ```
   Revision 'Rev.A' deleted in layout 'Master BOM'
   ```

   Or

   ```
   Revision 'Rev.A' deleted in 2 layouts
   ```

Delete All Revisions

Instead of deleting each revision one by one, the user can simply delete all revisions. The BOM tables will not be restored to the original state, but will be left as they currently appear (except that all Revision columns are reset to a blank value)

To delete a revision

1. Choose SC Assembly > BOM Revisions > Delete All BOM Revisions in Current Layout (page 500) or SC Assembly > BOM Revisions > Delete All BOM Revisions in All Layouts (page 500)
2. All revisions are deleted

   ```
   3 revisions deleted in layout 'Master BOM'
   ```

   Or

   ```
   3 revisions deleted in 2 layouts
   ```

List Revisions

Information about the revisions in the current layout can be printed to the command line. Revisions are listed with their name, description, user and date/time of creation.

To list revisions

1. Choose SC Assembly > BOM Revisions > List BOM Revisions (page 500)
2. Choose R (or enter) to display the revisions

```
Select what to list [Revisions/Operations/All]<Revisions>:
```

3. Choose N (or enter) to avoid seeing a long list of parts changed per revision.

```
List parts? [No/Yes]<No>:
```

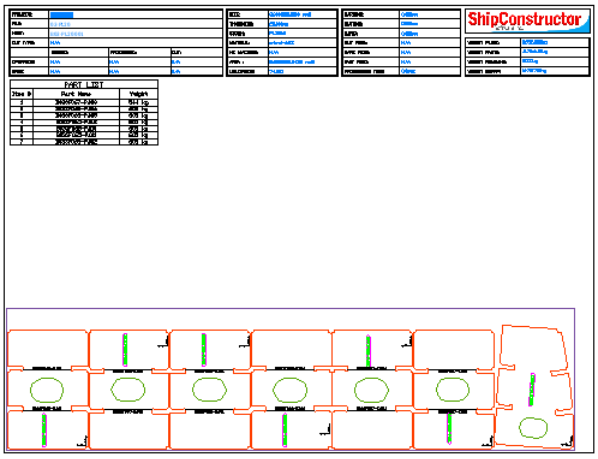
4. The revisions are then listed at the command line

Plate Nesting

Plate nesting is the process of arranging plate parts on stock plates or stock plate remnants in preparation for cutting.

A nest plate is a stock plate with various parts arranged on it. A nest consists of a nest plate, a BOM, and header fields with information like plate size, material, use, cutting time estimates, and so on.

A nest drawing is a ShipConstructor drawing that contains one or more nests of the same stock (that is, thickness and material). Each nest can have a different plate size and can include remnants.



A typical nest

Nesting Setup

Set Up Nest Options and Colors

Cut, marking, and no-process objects were given specific colors during the modeling phase. To distinguish if a part is assigned to a nest or not, unassigned parts are given different colors. When a part is placed inside the stock plate it will change back to the modeling colors.

To set up nest options

1. Choose ShipConstructor > Manager (page 383) to open Manager.
2. Choose General > Project Settings (page 327) to open the Project Settings window.
3. Open the section Project > Structure > Structure Colors > Nest Part Colors. This is where the unassigned colors are defined. The section Project > Structure > Structure Colors > NC-Pyros Colors is where the colors used during the exporting of nests are set.

4. Set the options (see General > Project Settings (page 327)).
5. Click OK to close the Project Setting window.

Set Up a Nest Naming Convention

You can control how ShipConstructor automatically names nests using naming conventions. See Naming Conventions (page 35). Nesting requires a naming convention.

To set up a nest naming convention

1. In Manager, choose General > Naming Conventions (page 336).
2. In the Naming Conventions window, select Structure > Nests > Plate and then click New. See Create a Naming Convention (page 38) for further details.

   Note: If you want the Unit name in the nest name use the Database Item called File Path and create the nest drawings in a Nest subfolder named after the Unit.

3. Activate the desired plate nesting naming convention. See Activate a Naming Convention (page 40).

Set Up NC Machines for Nesting

NC machines cut plates from plate stock. You must set up your NC machines before plate nesting (see NC Machines (page 90)).

Set Up Stocks for Nesting

Each stock has several properties that relate to nesting and NC cutting (for example, feed rate for cutting, cutting machine, nest part offset, nest edge offset, and stock plate sizes). See Create a Stock (page 51) and Plate Stock Properties (page 52) for details.

Set Up a Nest BOM Definition

The nest BOM will be a component of the nest that lists the parts in the nest.

To create or edit a BOM definition for nests

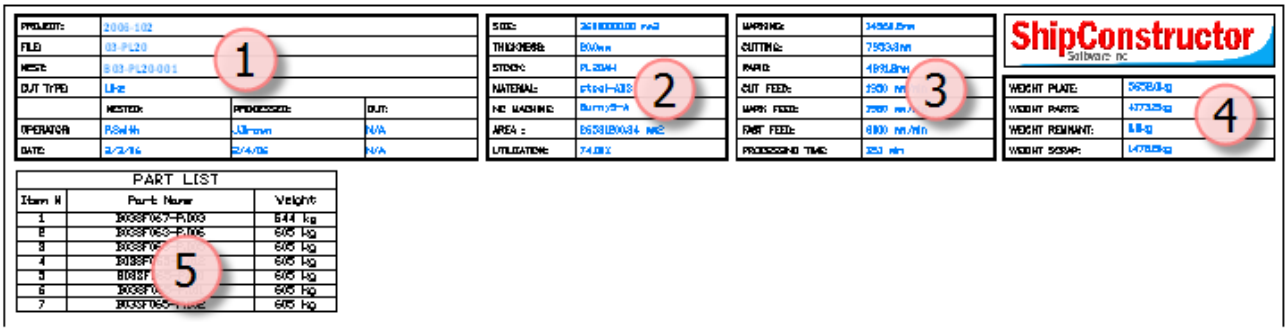
1. In Manager, choose General > Product Outputs > Bill Of Materials to open the BOM Manager.
2. Select the Structure > Nest node in the BOM Definitions list.
3. Click New. A new row under Nest is created.
4. Give the new BOM Definition a name.
5. Enter the Title of the Table. This will appear as a header row in the table.
6. Enter the AutoCAD Table style name that you want to use. This name should match with table style name that you will create in the template drawing.
7. Select a label style. The label style is used for labeling items in the viewport with the item #.
8. Set BOM Field options in the BOM Definitions Manager (page 47)

Set Up Nest Templates

A nest template controls the layout and appearance of each nest within a nest drawing. ShipConstructor includes a default nest template that you can customize, or you can create your own nest templates.

The top of the default template contains several fields for project name, nest name, and so on. To the right of each field is a keyword (for example, <PROJECT NAME>). When you create nests, ShipConstructor automatically replaces each keyword with the appropriate information. You can move keywords, change their size or color, or delete them if you do not need them. You can also insert additional keywords into the template, and they will appear in all nests (see Insert Keywords into a Nest (page 281)).

Each nest template must contain exactly one template border, one BOM table, and one <NEST STOCK PLATE LOCATION> keyword to be valid for nesting.



A typical nest template header with information areas

Area 1 – Contains general information about the nest, such as the names of the project, the drawing file and the nest. The cut type indicates if the plate will be cut as Like or Like & Mirror. Below this are the operator names and dates for nesting, NC processing (with NC-Pyros), and cutting. The process and cut information is not entered yet, as these procedures have not yet been completed.

Area 2 – This area contains the nest information, such as plate size, thickness, stock, material, area, and use.

Area 3 – This area contains the NC processing information, such as the length of cutting, marking and rapid travel, the feed rates, and the projected cutting time. These values will only be available after the plate has been processed by NC-Pyros.

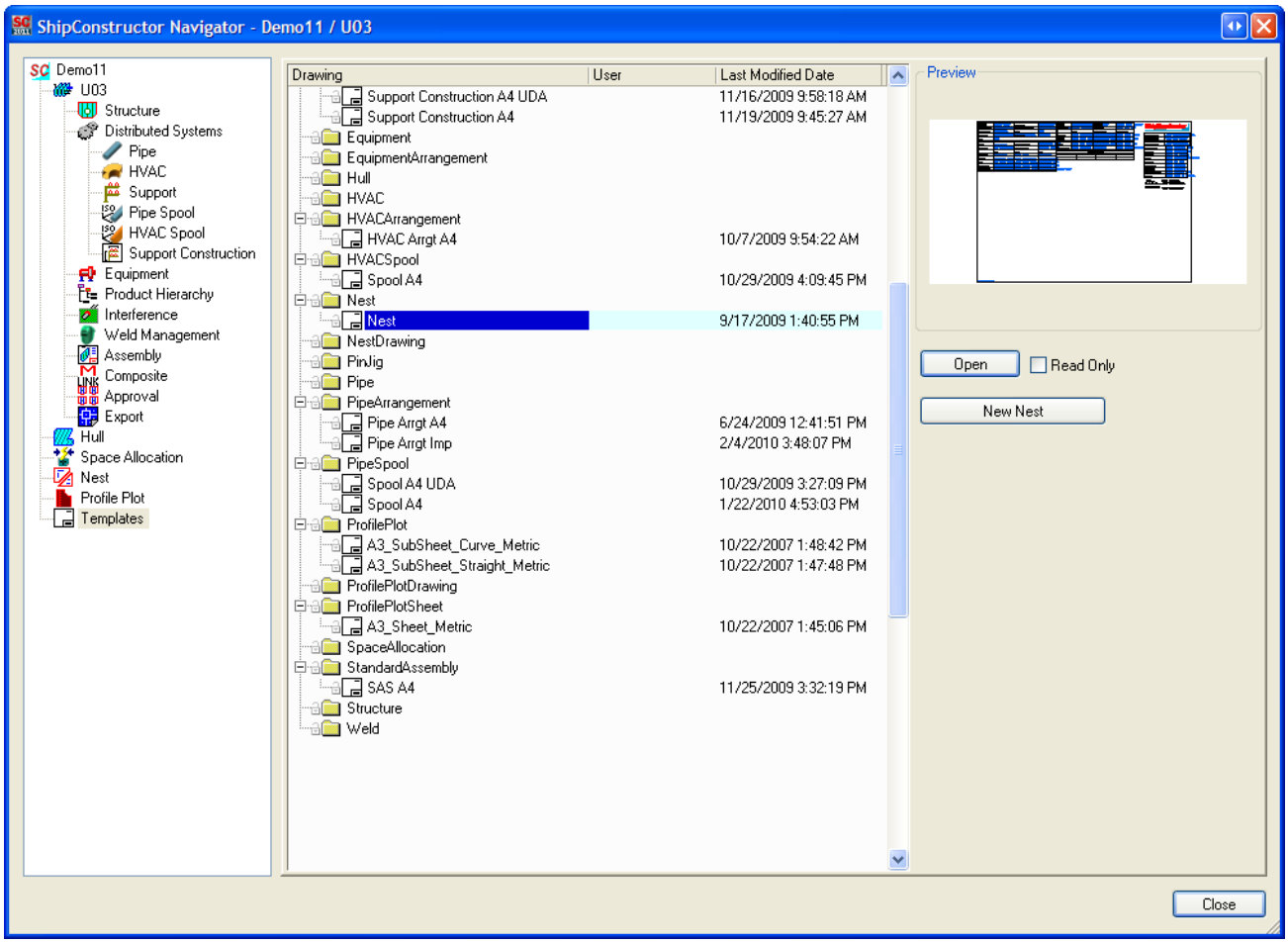
Area 4 – This area shows the weight of the plate, weight of the parts, weight of the remnants if used, and the weight of the scrap.

Area 5 – This area contains the Bill of Material. As shown before, the BOM is configured using Manager (see Set Up a Nest BOM Definition (page 255)).

Open a Nest Template

To open the default nest template

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.
2. Select the Templates page.
3. In the Nest folder, select the Nest drawing.



4. Click Open.

Create a Nest Template

To create a nest template

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.
2. Select the Templates page.
3. Select the Nest folder.
4. Click New Nest.

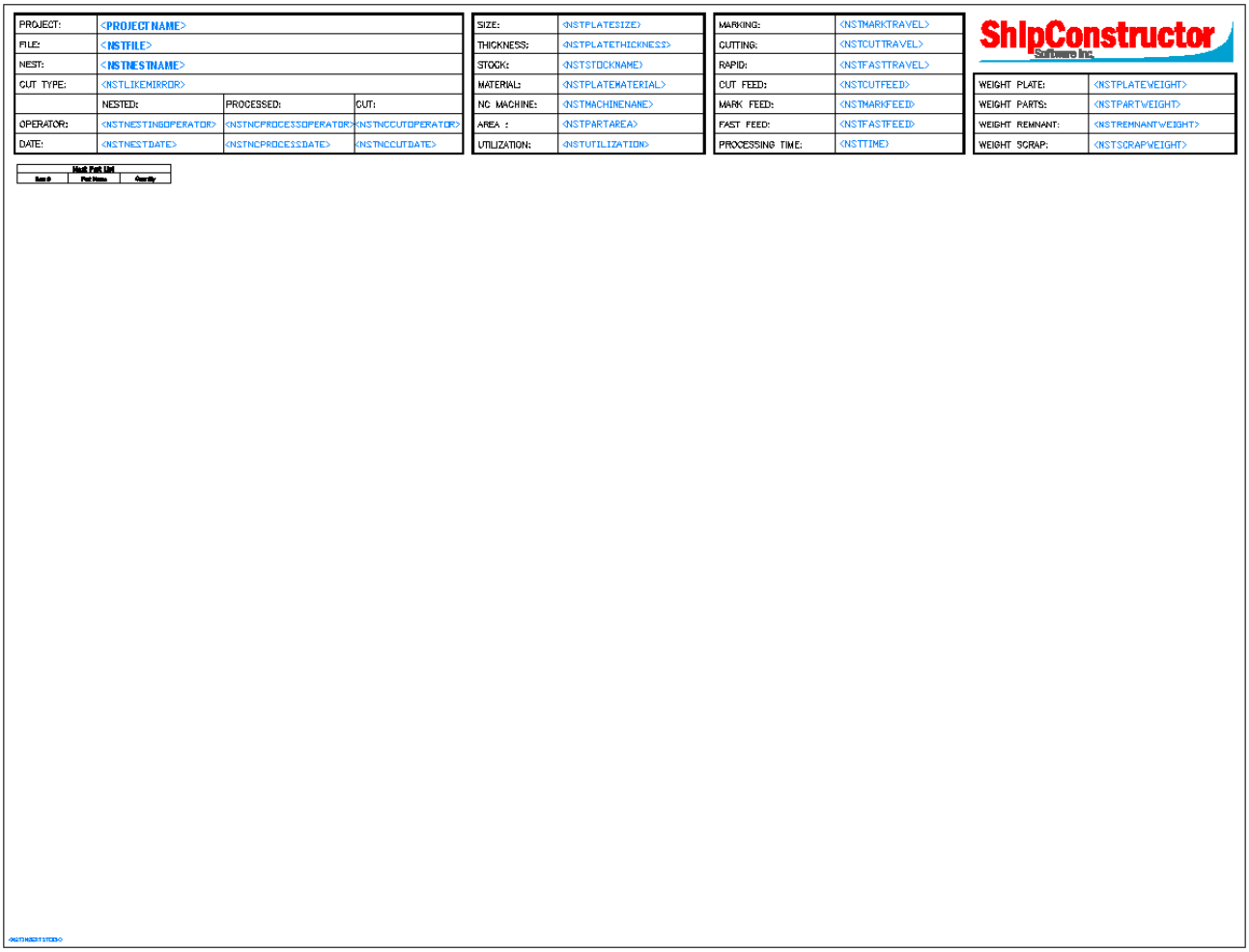
   The New Drawing window appears.
5. Enter a name for the nest template.
6. Click OK to close the New Drawing window.

Define a Nest Template Border

A nest template must contain one border or frame. The border consists of a closed polyline, usually a rectangle. ShipConstructor uses this border to determine which parts and other objects (such as the BOM or keywords) are part of a specific nest. Nest templates from previous versions (ShipConstructor2005 and earlier) need to be updated before they are used by ShipConstructor.

To define a nest template border

1. In the nest template, choose SC Plate Nest Template > Define Border (page 526).
2. Select a closed polyline to use as the border.



To update a nest template

1. Choose SC Plate Nest Template > Update from a Previous Version (page 529).
2. Select the existing template border closed polyline that you want to update.
3. Confirm that all objects are on the stock layer.

Note: The BOM keyword from previous versions is no longer used, so it can be deleted.

Insert Keywords into a Nest Template

Nest templates can contain any number of keywords. Each nest template must also contain the keyword <NEST STOCK PLATE LOCATION> to indicate the location of the nest plate. ShipConstructor places the lower left corner of the nest plate or remnant at the insertion point of the keyword. Without this keyword, nesting will fail.

Keywords use three different terms for various properties: Plate Stock is the original raw stock for the nest. Nest Stock is the item used to create the nest (either a remnant or raw stock). Nest is the current nest (the nest stock excluding all remnants on the nest).

To insert keywords into a nest template

1. Choose SC Plate Nest Template > Insert Keywords (page 526) to open the Insert Keywords window.
2. Select the keyword.
3. Set the Text Style, Text Size, and Text Rotation.
4. Click OK.
5. Select the insertion point for the keyword.

Check Drawing Validity

Nest template drawings require objects to exist in the drawing before the drawing can be properly used in nesting. These objects are: a single template border, a single <NEST STOCK PLATE LOCATION> keyword, a single BOM table. While it is not necessary, it is suggested that the border be the largest object in the drawing, encompassing everything else.

To check a nest template

1. Choose SC Plate Nest Template > Check Drawing (page 529).
2. A check will run, confirming that the criterion is met.
3. A message window will appear listing any errors, if any.
4. If there are no errors, then a check is run, finding all objects outside the bounds of the border. A message window will appear listed how many of these objects are found, and the option to select them.

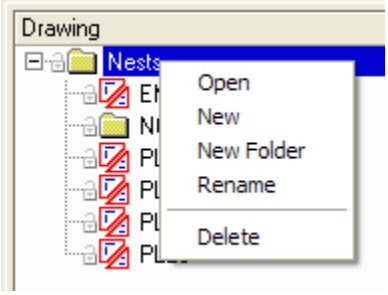
Set Up Nest Drawings and Nests

Create a Nest Folder

You may want to create folders within the Nest folder in order to organize your nest drawings.

To create a nest folder

1. Choose ShipConstructor > Navigator (page 368) to open Navigator and select the Nest page.
2. Right-click in the Drawing list and choose New Folder.



3. Enter a name for the folder.

Note: The name of the folder can be anything, but for better organization, it is usually a Unit name, and all nest drawings for that Unit are placed in the folder.

Create a Nest Drawing

Each nest drawing can contain one or more nests of the same stock but can have different plate sizes, including remnants. Many companies decide to use the stock name as the drawing name. Often, the unit, that most or all of the parts on the nest belong to, is also integrated into the drawing name. For example, U12PL10.DWG contains parts for U12 on PL10 stock plate. You can organize your nests by unit by creating folders for each unit.

Sometimes nesting cannot be carried out on a unit-by-unit basis. You might be building more than one unit at a time. Nesting just one unit can also create too much unused plate space. The unused space can be treated as a remnant or

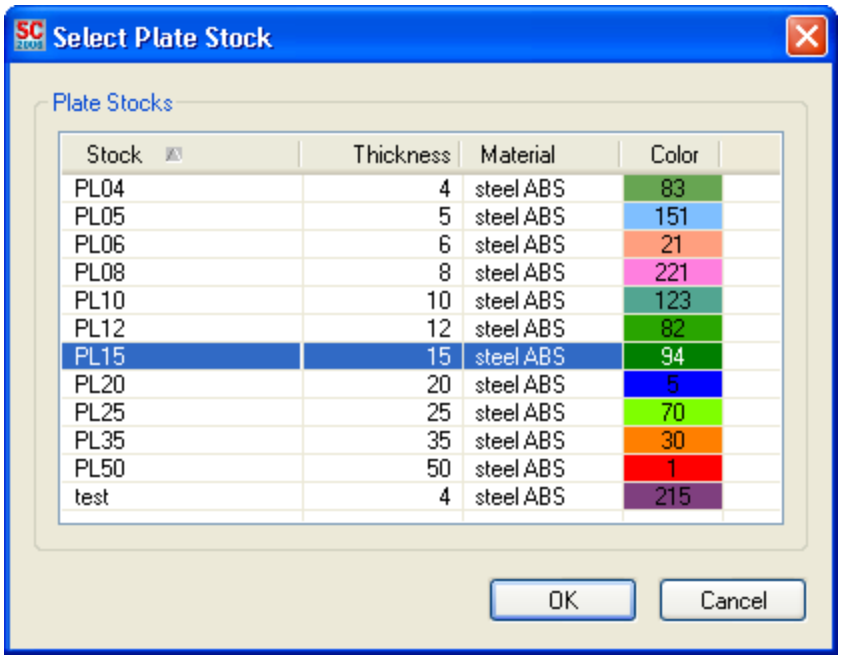
drop. A remnant is the unused portion of a stock plate that you can use to create a new nest (see Set Up Remnants (page 285)).

To create a nest drawing

1. Choose ShipConstructor > Navigator (page 368) to open Navigator and select the Nest page.
2. Select a folder in the drawing list.
3. Click New.

   The New Drawing window appears.
4. Enter a name for the nest drawing.
5. Click OK to close the New Drawing window.

   The Select Plate Stock window appears.



6. Select a plate stock for the nest drawing.
7. Click OK to close the Select Plate Stock window.

Re-create a Nest Drawing

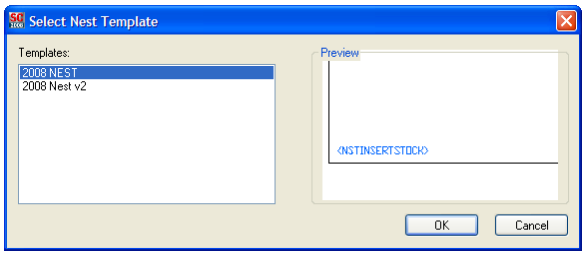
Each nest drawing can be re-created using the data stored in the database, similar to model drawings. Older nest drawings will be partially re-created. Their parts will be placed within the nest they are currently assigned to, but the original location and orientation will have been lost.

Un-nested parts will be placed off to the side.

To re-create a nest drawing

1. Choose ShipConstructor > Navigator (page 368) to open Navigator and select the Nest page.
2. Select a folder in the drawing list
3. Select the nest drawing in the list. Click Open.

   A message box will appear prompting to re-create the nest drawing from the database. Click Yes to re-create the drawing.
4. If there is more than one nest template drawing defined, then the Select Nest Template window will appear. Select the nest template to use for re-creating the nests in the drawing. Click OK to continue.

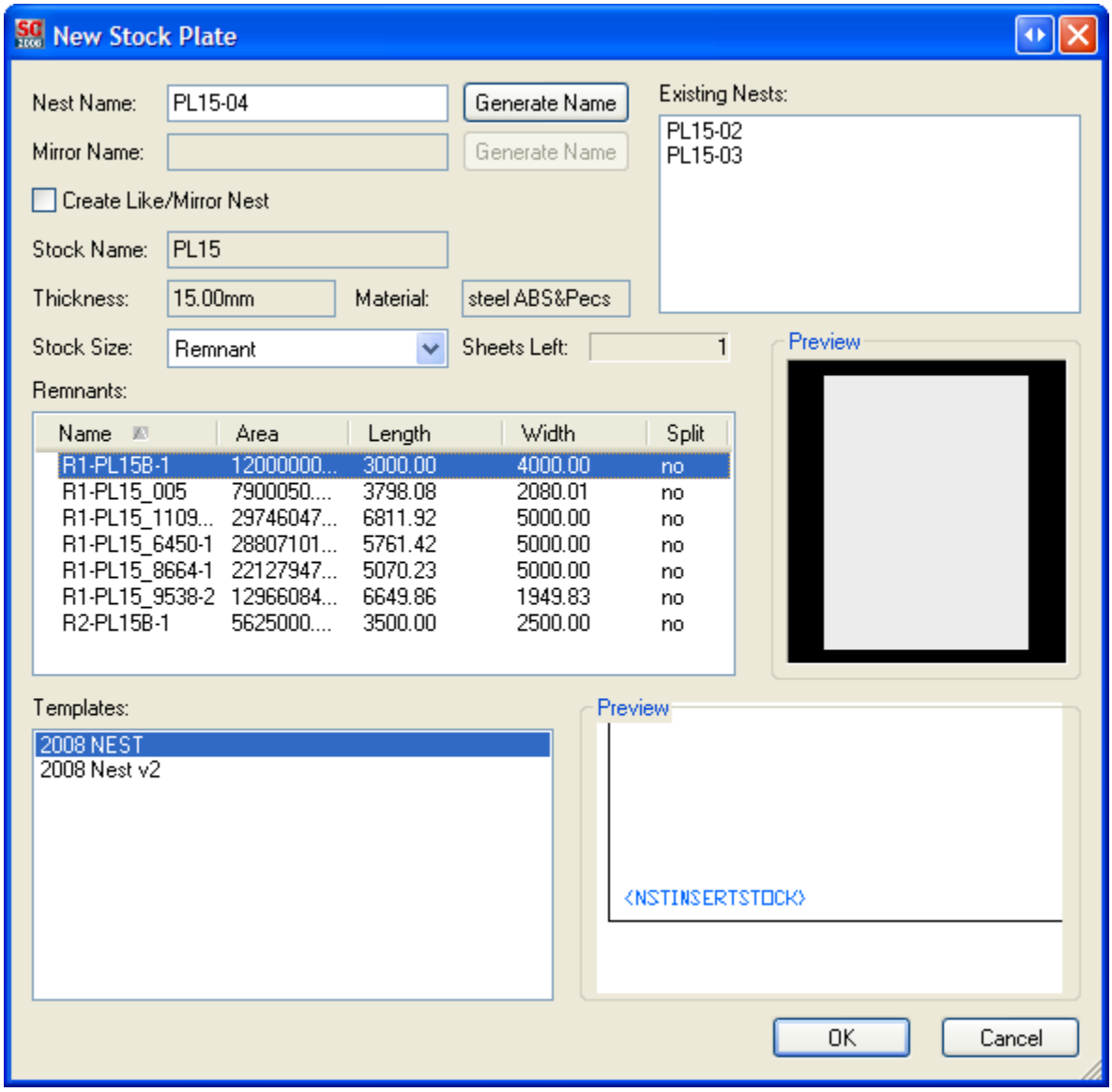


Insert a Nest into a Nest Drawing

Note: If you have a license for automatic nesting, you do not need to manually insert a nest into a nest drawing. You can simply insert and arrange parts into a nest drawing (see Automatically Insert and Nest Several Parts (page 266)).

To insert a nest into a nest drawing

1. Choose SC Plate Nest > Nest > New (page 511) to open the New Stock Plate window.



2. Set the options.
3. Click OK.
4. Click in the drawing to set the location for the new nest.

   ShipConstructor automatically names the nest. For more information see SC Plate Nest > Nest > New (page 511).

Insert Parts into a Nest

There are three ways you can insert parts into a nest:

- Inserting one part at a time
- Arraying several parts at once
- Automatically insert and nest several parts (requires a license for AutomaticNest)

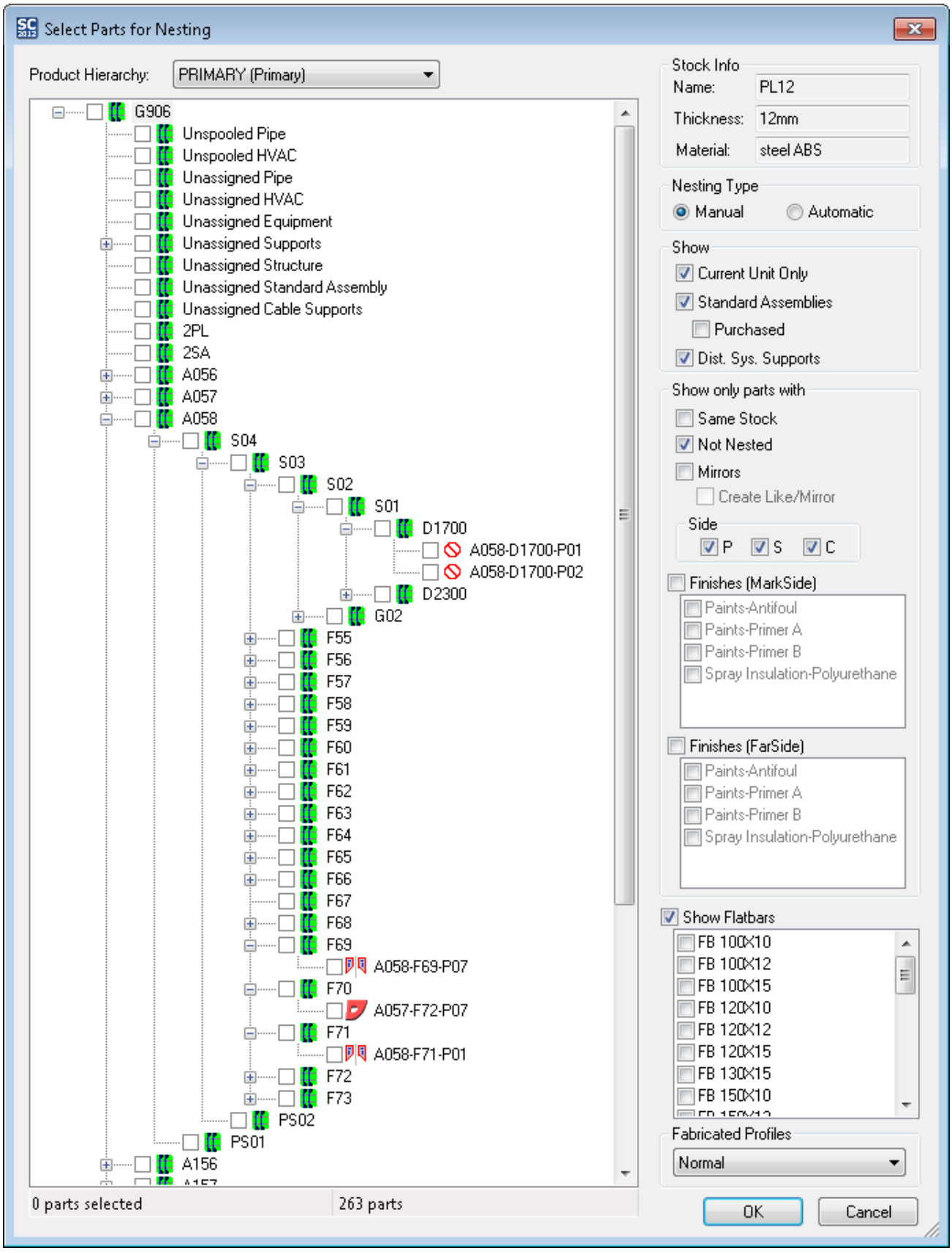
Keep in mind the following when arranging parts on a plate:

- Insert the largest part first, then the second largest, and so on.
- Use like and mirror cuts. A like and mirror plate will only be processed once, but cut twice, reducing processing time by half.
- Arrange parts onto nests based on the sequence of production. Keep parts that will be needed at the same time on the same nest.

Insert Parts Into a Nest One at a Time

To insert parts into a nest one at a time

2. Choose SC Plate Nest > Part > Insert (page 514)
3. The Select Parts for Nesting window appears.


Select Parts for Nesting
Product Hierarchy:
PRIMARY (Primary)
G906
Unspooled Pipe
Unspooled HVAC
Unassigned Pipe
Unassigned HVAC
Unassigned Equipment
Unassigned Supports
Unassigned Structure
Unassigned Standard Assembly
Unassigned Cable Supports
2PL
2SA
A056
A057
A058
S04
S03
S02
S01
D1700
A058-D1700-P01
A058-D1700-P02
D2300
G02
F55
F56
F57
F58
F59
F60
F61
F62
F63
F64
F65
F66
F67
F68
F69
A058-F69-P07
F70
A057-F72-P07
F71
A058-F71-P01
F72
F73
PS02
PS01
A156
0 parts selected
263 parts
Stock Info
Name:
PL12
Thickness:
12mm
Material:
steel ABS
Nesting Type
Manual
Automatic
Show
Current Unit Only
Standard Assemblies
Purchased
Dist. Sys. Supports
Show only parts with
Same Stock
Not Nested
Mirrors
Create Like/Mirror
Side
P
S
C
Finishes (MarkSide)
Paints-Antifoul
Paints-Primer A
Paints-Primer B
Spray Insulation-Polyurethane
Finishes (FarSide)
Paints-Antifoul
Paints-Primer A
Paints-Primer B
Spray Insulation-Polyurethane
Show Flatbars
FB 100X10
FB 100X12
FB 100X15
FB 120X10
FB 120X12
FB 120X15
FB 130X15
FB 150X10
Fabricated Profiles
Normal
OK
Cancel

The left side of the window displays the assembly tree with all parts that match the filter setting. The assembly tree makes it easy to nest by construction unit or any other assembly. If you click the check box that is in front of an assembly, all sub-assemblies and parts are automatically selected. If you change the value in the Product Hierarchy dropdownlist, the assembly tree updates to show the assemblies for that product hierarchy.

The bottom of the window confirms the number of parts selected for nesting.

Stock Info displays the stock properties used in this nest drawing.

Nesting Type is the options for the type of nesting to perform: Manual or Automatic.

On the right side of the window, Show lists various grouping options to display. Below it, Show only parts with lists the controls for filtering the parts. Only parts that match the selected options will be available for nesting. In this case there are the following settings:

- Current Unit Only – This option minimizes the amount of information in the tree, speeding up the window, and is especially useful for large projects. This only applies to the primary product hierarchy. For non-primary product hierarchies the entire project is shown.
- Standard Assemblies – If checked, the filter will display assemblies from standard assemblies, and their parts.
- Purchased – If checked, Standard Assemblies tagged as “Purchased” will also be displayed.
- Dist. Sys. Supports – If checked, Distributed Systems Supports and their parts are displayed.
- Same Stock – Only parts using the same stock should be nested in one nest drawing. Uncheck this only if you want to force a part to be nested on another stock, for example, 8 mm brackets onto 10 mm stock or nesting a part made from corrugated stock onto plate stock.
- Not Nested – If checked, the filter will only display parts that have not already been placed in nest drawings.
- Mirrors – Only mirrored parts are listed.
- Create Like/Mirror Nest – Only enabled when Mirrors is selected. Choose this option to create like-and-mirror nests (using mirrored parts). A like-and-mirror nest will only be processed once, but cut twice. Many cutting machines can do this at the same time, reducing processing time by half
- Side – You can filter by the side of the vessel the parts are on.
- Finishes – If checked, this lets you select a finish option. For example, you might have some primed plate and only want to nest parts on these that are to be cut from primed plate. The MarkSide and FarSide options allow you to filter parts that only have the selected finish on the chosen side.
- Show Flatbars – If this option is selected, the flatbars listed in the window below this option can be selected to appear in the nesting tree. Flat bars of more than the present stock are shown in case you decide that ordering more stock of a certain type is not a good use of resources. So you can choose to have some flat bars made out of a stock that you want to use up, instead of ordering more of another type that you may have run out of.

Note: Flatbar parts will be unrolled or expanded when they appear in the nest drawing.

- Fabricated Profiles: - Determines how fabricated profile components are displayed in the nest drawing.
  - Normal – The profile component will appear similar to how it is in the model drawing. It will contain cutout, endcut, and trim details.
  - Straightened – Undetailed – The profile component will be a rectangular shape with no cutouts, endcuts, or trims.
  - Straightened - The profile component will be a rectangular shape but also contain cutouts, endcuts, and trims.

4. Select the parts to nest. Click OK.

   The Nesting Parts Insertion Options window appears. This window gives you additional options for manual nesting.

5. Select “Insert Parts Individually” and Click OK.
6. Insert parts.

Note: The largest part is inserted first, the smallest one last. The sorting algorithm selects the part it considers largest first. So, as you start to nest, you will be given the largest part first.

Note: Be sure to set the OSNAP OFF and ORTHO OFF. Otherwise you might not see the part at all.

The general procedure for inserting parts into a nest drawing is as follows:

Parts are inserted at the mouse position, automatically rotated to the smallest horizontal rectangle, with the move command active.

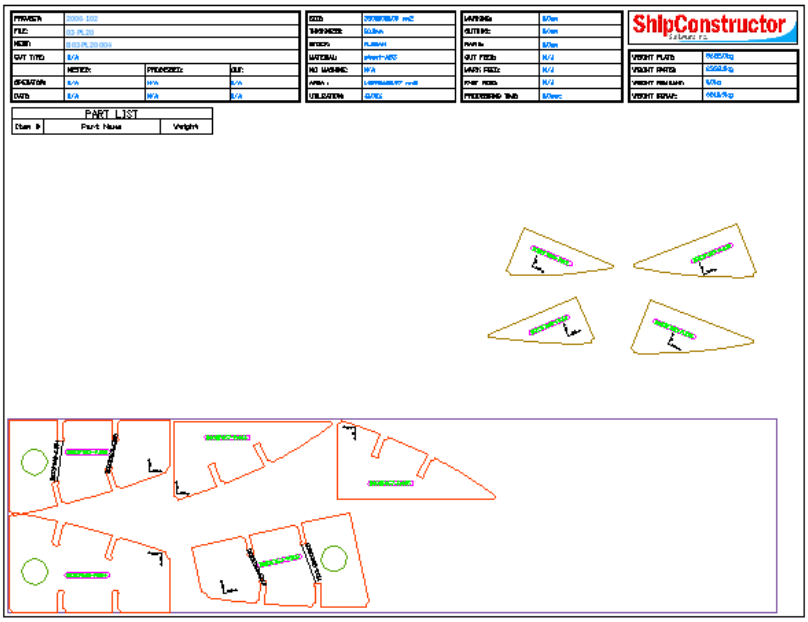
Move the mouse to move the part.

You can change the rotation using the left and right arrow keys. Use the plus (+) and minus (-) keys to increase or decrease the rotation step-size.

7. The first part is displayed at the mouse position. ShipConstructor automatically rotates the part to the best horizontal position. The command prompt reads:

```
Inserting 0301-P07-P02:
Rotate with arrow keys, Modify the step-size with +/- <15>
```

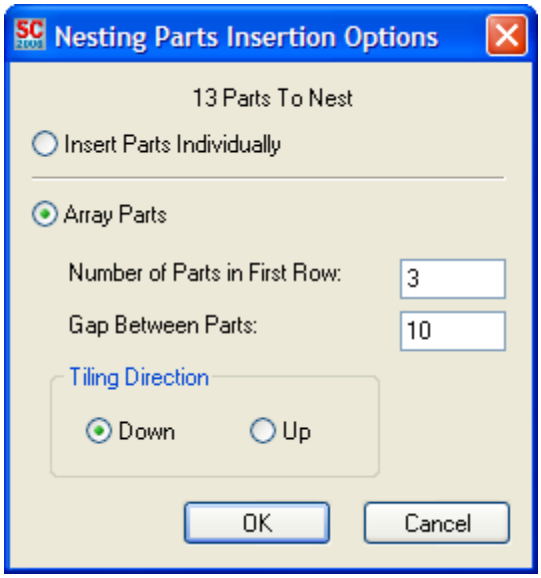
8. Place the parts in an organized way in the nest drawing for later nesting. The figure shows an organized placing of parts.



Array Several Parts at Once Into a Nest

To array several parts at once into a nest

1. Choose SC Plate Nest > Part > Insert (page 514)
2. The Select Parts for Nesting window appears.
3. Set the options. See Insert Parts Into a Nest One at a Time (page 262).
4. Select the parts to nest and click OK.
5. The Nesting Parts Insertion Options window appears with additional options for manual nesting.



For the Number of Parts in First Row, you need to consider the sizes of the largest parts that will be inserted. The largest parts will be inserted first. Since the first parts are the largest, it is usually good to choose a small number of parts for the first row.

- Gap Between Parts -- The distance that should exist between the parts if you want to add the required spacing for the cutting operation.
- Tiling Direction -- The direction, starting on the left side, in which you want the array of parts to go.

Note: The largest part is inserted first, the smallest one last. The sorting algorithm selects the part it considers the largest part first. This sometimes produces unexpected results with the Array option due to some parts having odd boundaries.

Note: Be sure to set the OSNAP OFF and ORTHO OFF. Otherwise you might not see the part at all.

The general procedure for inserting parts into a nest drawing is as follows:

If you have selected the Down option for the Tiling Direction, the parts are inserted with the mouse position being the top left of the array of parts. If you have selected the Up option for the Tiling Direction, the first part is inserted as the bottom left of the array of parts.

Move the mouse to move the part into the position where you want to start the array.

- The rest of the parts will be added according to the choices you have made and the algorithm's decision of the descending size order of the parts.

Note: Some parts have unusual boundary properties and will not be inserted with the proper spacing. Most will work as expected. This is just a helper function to aid in nesting to get the parts quickly onto the screen. For true automated nesting, you will need the AutomaticNest module.

6. The first part is displayed at the mouse position. ShipConstructor automatically rotates the part to the best horizontal position. Assuming you have selected the Tiling Down option, the command prompt will read:

```
Insert the first part at the top left for the starting position of the array.
```

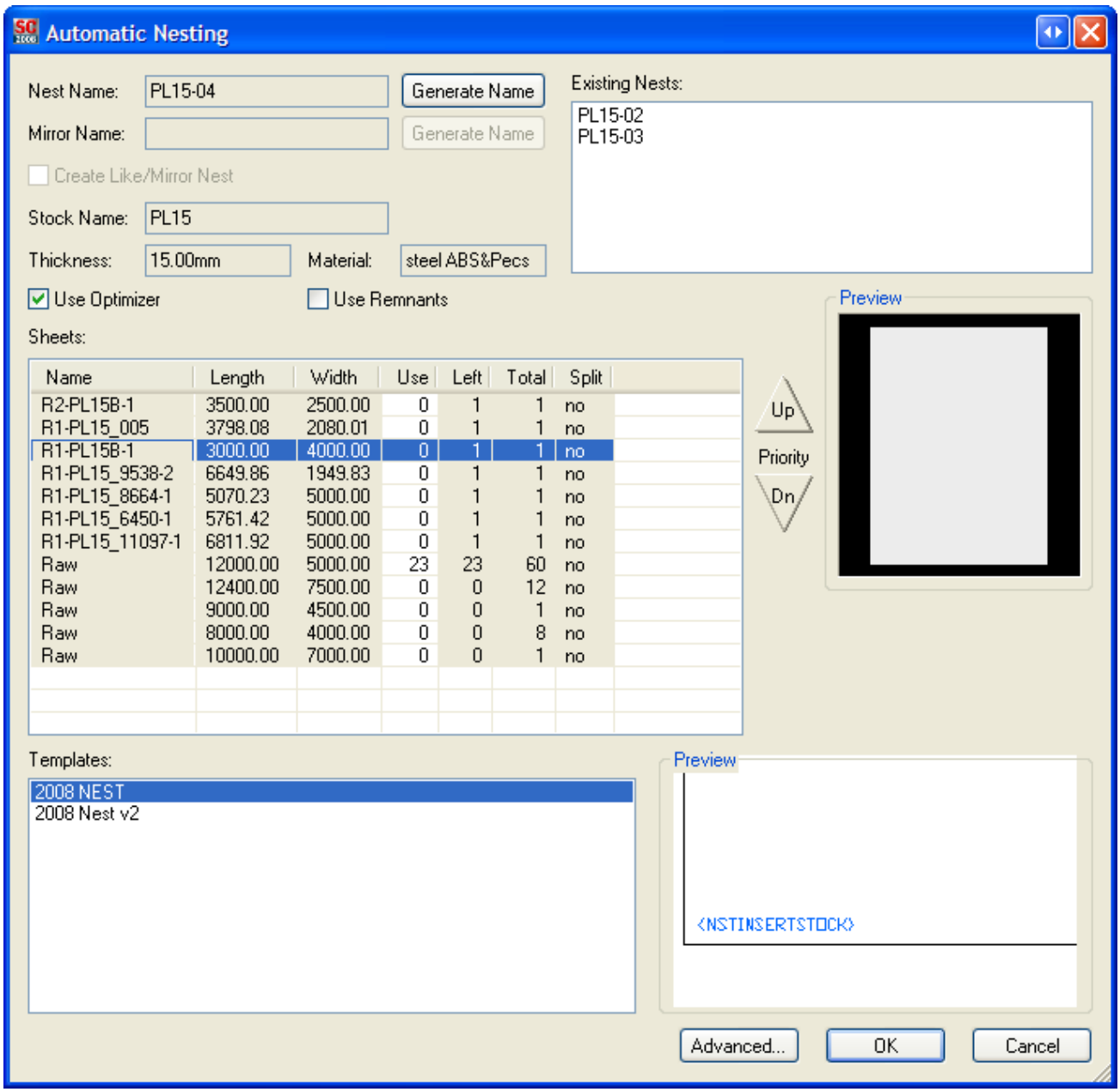
7. Once inserted, the rest of the parts are added automatically. You will probably need to manually place the parts into their final positions. Additional functions for aligning and moving parts are listed further on in this chapter.

Automatically Insert and Nest Several Parts

To automatically insert and nest several parts

1. Choose SC Plate Nest > Part > Insert (page 514)
2. The Select Parts for Nesting window appears.

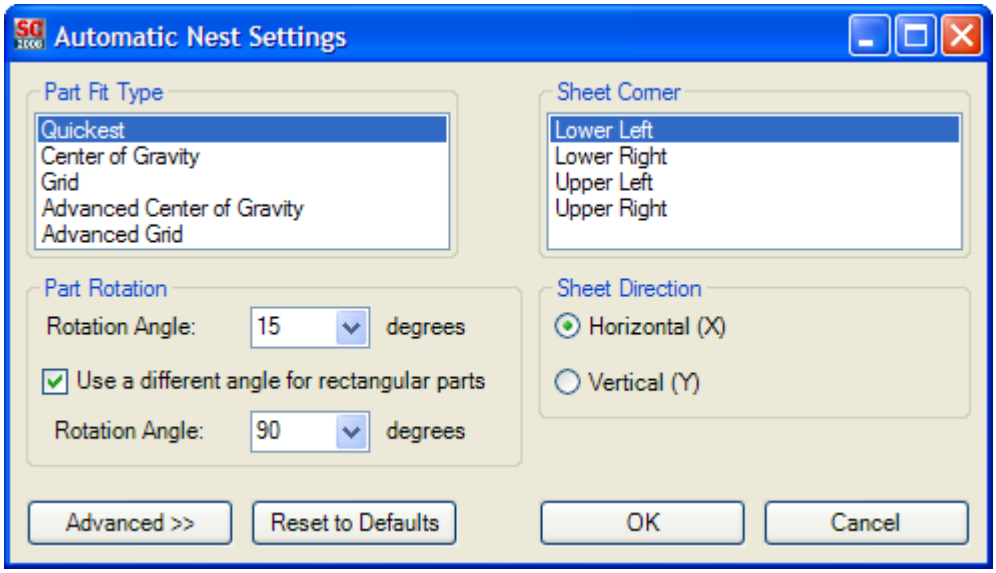
3. Select Automatic for Nesting Type.
4. Set the desired options. See Insert Parts Into a Nest One at a Time (page 262).
5. Select the parts to nest and click OK. ShipConstructor now inserts the blocks for all the parts you selected into the drawing at the origin.
6. The Automatic Nesting window appears.



- Nest Name – Automatic nesting creates nests. The nests will be named with the given name, and if that name exists, automatic nesting will use the next number.
- Mirror Name – The name of the mirror nest (if creating Like and Mirror nests).
- Stock Name – The name of the stock assigned to the current drawing.
- Thickness – The thickness of the stock assigned to the current drawing.
- Material – The material of the stock assigned to the current drawing.

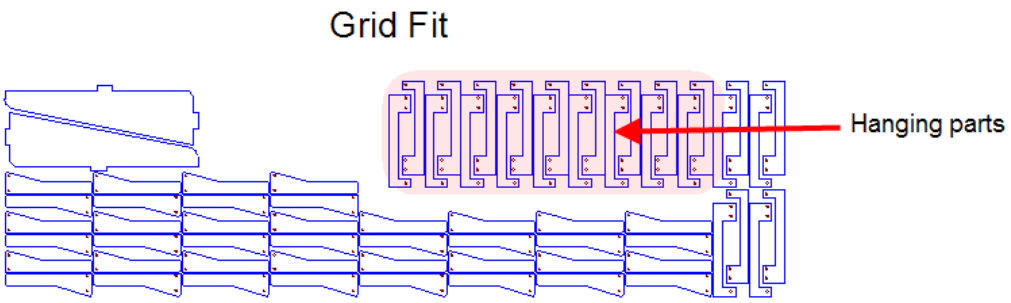
- Use Optimizer – Enables the Optimizer for the nesting operation.
- Use Remnants – Toggles the “Use” count for remnants in the Sheets list.
- Existing Nests – A list of the existing nest names for the project.
- Sheets – A list of the available sheets to use. Remnants are always listed before raw stock, with the smallest remnant first. Sheets at the top of the list will be used before those lower down on the list. If parts cannot fit on a given plate size, the next size down is tried. You can reorder the sheets by selecting a size in the list and clicking the Up or Dn buttons. The order of the list is initially taken from the plate sizes set up in Manager. See Edit Sizes Window (page 343).
- Templates – A list of template drawings. Select the template that you want to use.
- Advanced – Shows the advanced settings for automatic nesting (Recommended).

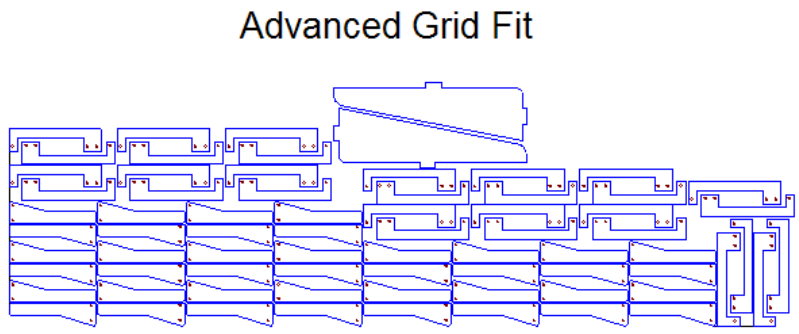
7. Click Advanced to see and set the options in the Advanced Nest Settings window.



Part Fit Type – Automatic nesting contains different algorithms for how parts are nested:

- Quickest – This type of fit provides the fastest result but seldom provides the best results.
- Center of Gravity – The center of gravity of the part is positioned depending on the Preferred Nest Direction.
- Grid – This is a good choice of algorithm for rectangular style parts.
- Advanced Center of Gravity – Similar to Center of Gravity.
- Advanced Grid – Similar to Grid except hanging part groups are handled.





Part Rotation – The step angles to rotate the parts when trying to place it. The parts start at an angle that creates the minimum height. The smaller the angle, the longer it takes to complete the nesting.

Rotation Angle – Angle step size for parts.

Use a different angle for rectangular parts – The option enables another angle to be used by rectangular parts, instead of the smaller angle (such as 15°) used previously. Finding the best fit of a rectangular part is faster if a larger angle is used, such as 90°.

Rotation Angle – Angle step size for rectangular parts.

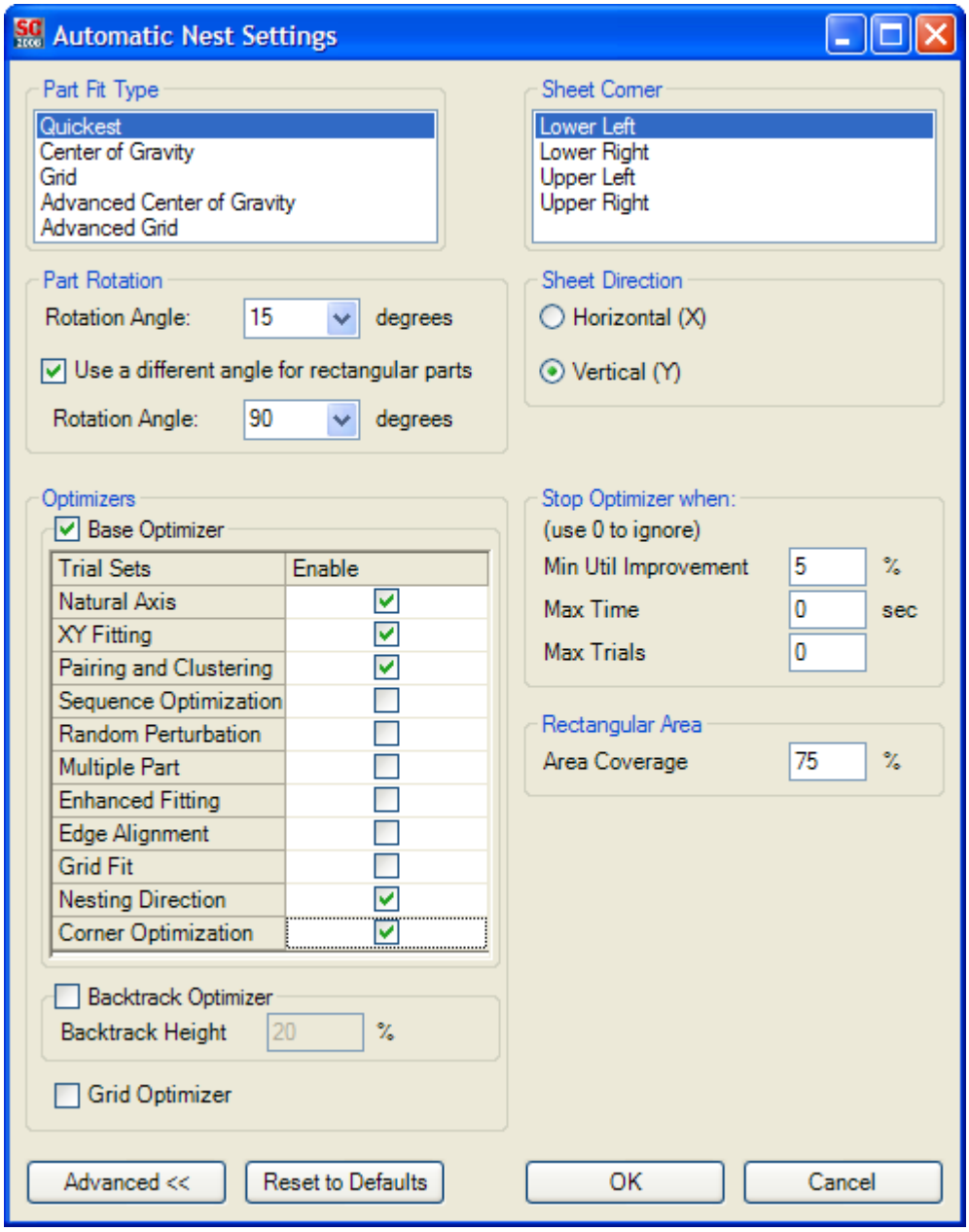
> Tip: When you are automatic nesting, you may find that the results are not acceptable. Testing different types of fit is a good way to ensure optimal nesting. A smaller rotation angle does not guarantee better use. A good starting angle to test with is 45°.

Sheet Corner – The corner of the stock plate that the parts are nested closest to.

Sheet Direction

- Horizontal (X) – Parts are placed beside each other first rather than on top of each other.
- Vertical (Y) – Parts are placed on top of each other first rather than beside each other.

Click Advanced to expand the window to show more advanced options, related to the Optimizer. These values do not need to be changed for most users, but if you feel the need to slightly adjust some settings, then they can be modified here.



Optimizers – The Optimizer works by generating the default nest output, using the automatic nest settings. This gives it a base case to improve by using various trials. Each trial is a different algorithm to use in achieving a better utilization over the default nest layout. Trials are grouped into trial sets, and each Optimizer contains different trial sets to add to the list.

- Base Optimizer – The Base Optimizer contains 11 trial sets, totaling 44 trials. Each trial set can be enabled or disabled in the grid.
- Backtrack Optimizer – The Backtrack Optimizer contains 30 trials in a single trial set. It uses a different group of algorithms over the Base Optimizer.
- Backtrack Height – The sheet height from where the backtracking starts.
- Grid Optimizer – The Grid Optimizer is designed to fix grids with hanging parts, to improve the layout. It contains 19 trials in a single trial set.

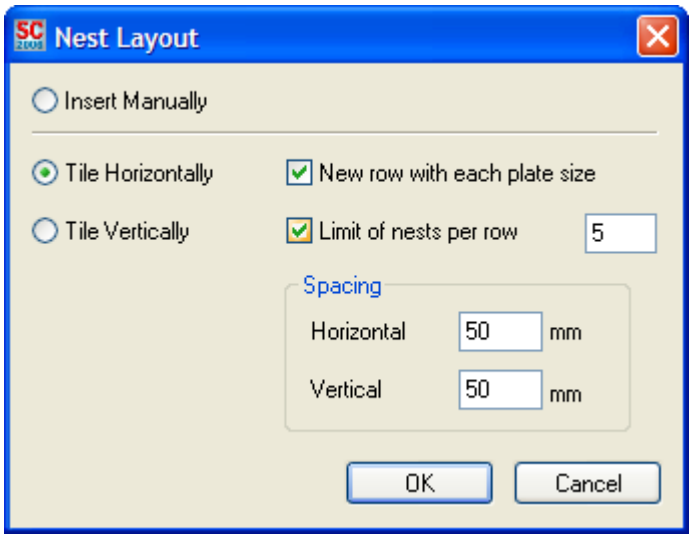
Stop Optimizer when – The Optimizer to attempt each trial, and after all selected have been used, the best layout is used. However, the user can specify various parameters to stop Optimizer before all trials have been run. Using 0 for any of these parameters will ignore it.

- Minimum Utilization Improvement – The default nest has the basic utilization. If any of the trials have a utilization that is this percent above that basic utilization, then stop the Optimizer and use that trial's layout.
- Maximum Time – The longest time that the Optimizer should run. Once this time has been reached, the Optimizer will stop, but only after the current trial has finished. This means, the Optimizer will run longer than this time, but only so that it can complete the current trial. The best layout found to date will be used.
- Max Trials – The sets the upper limit of how many trials to use. By default, the Optimizer will stop when there are no more trials to test. The user can set a smaller value here, and the best layout in the trials tested will be used.

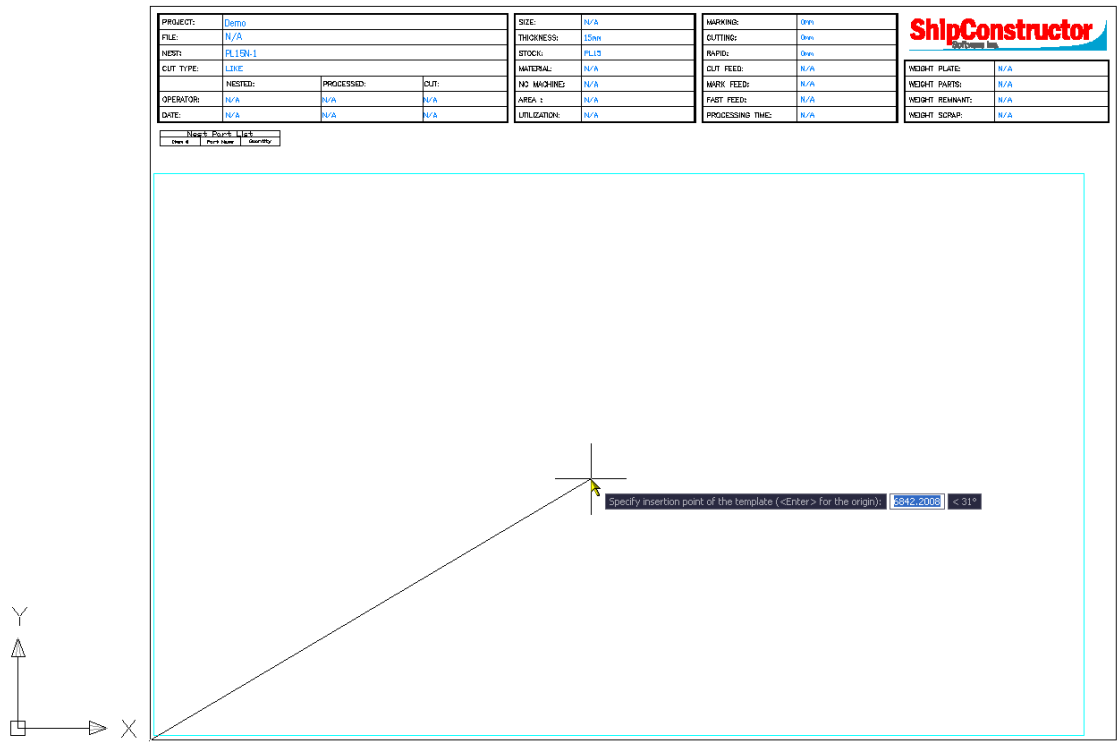
Click Reset to Defaults to restore all values to their values before this window was opened.

After setting up the advanced settings, click OK to return to the Automatic Nesting window.

The Nest Layout window appears. This window lets you determine how the new nests will be placed in the drawing.



- Insert Manually – If this is checked, you will choose the position of all the nest plates one at a time.
- Tile Horizontally – If this is checked, you will pick the location of the first nest and all subsequent nests will be tiled horizontally.
- Tile Vertically – If this is checked, you will pick the location of the first nest and all subsequent nests will be tiled vertically.
- New row (or column) with each plate size – Whether or not to start a new row or column for each plate size.
- Limit of nests per row (or column) – If this is checked, the number in the box will be the maximum number of nests used in a row or column before starting a new row or column.
- Spacing – How much to separate the nests by when automatically tiling them.



8. If you are tiling, choose the position of the first nest. If you are inserting manually, you will be prompted to pick the insert point for each nest.

Remove Parts from a Nest

To remove parts from a nest

1. Choose SC Plate Nest > Part > Remove (page 515)
2. Select the parts to remove.

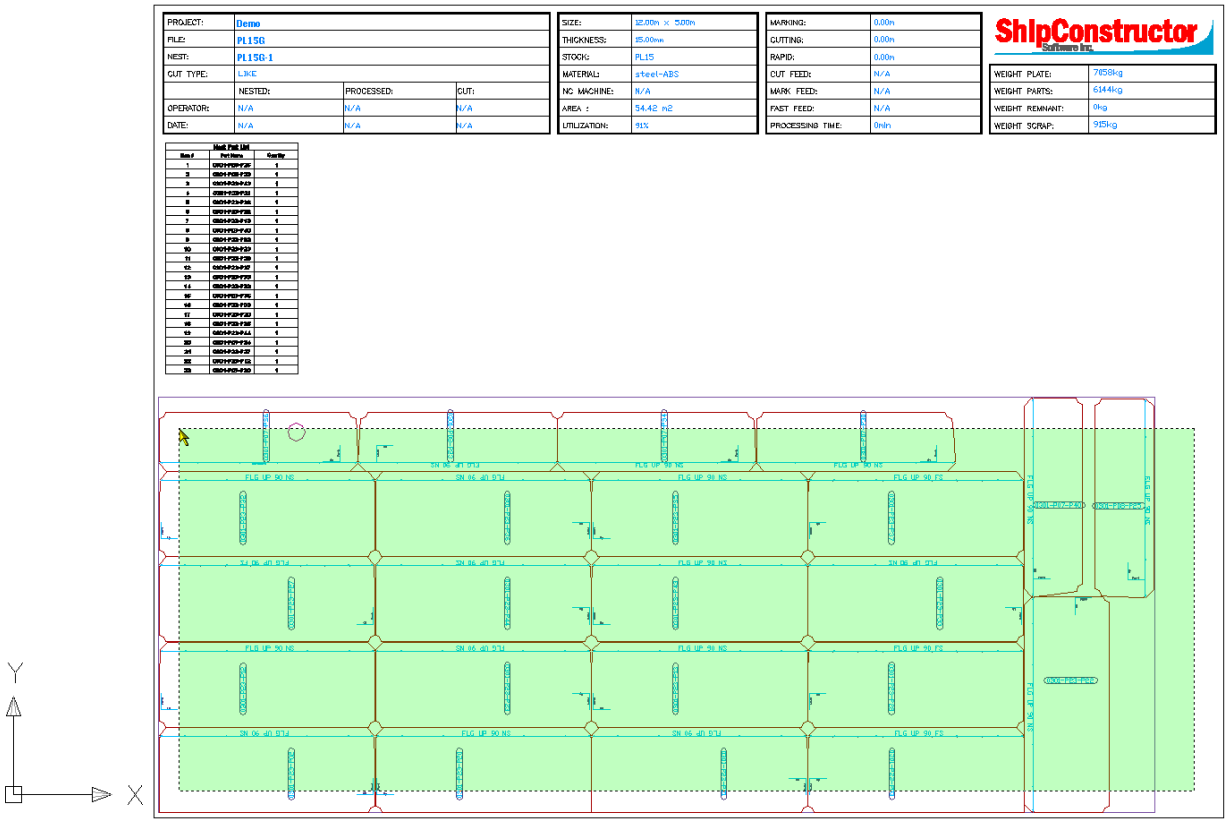
> Note: You can also use the AutoCAD **ERASE** command to un-nest a part.

Re-nest a Nest

You can apply a different nesting algorithm to a set of nested parts and automatically re-nest them (for example, if you are not satisfied with the original nesting).

To re-nest parts

1. Choose SC Plate Nest > Nest > Re-Nest (page 512)
2. Select the parts you want to re-nest.



3. The Automatic Nest Settings window appears, letting you make changes as needed.

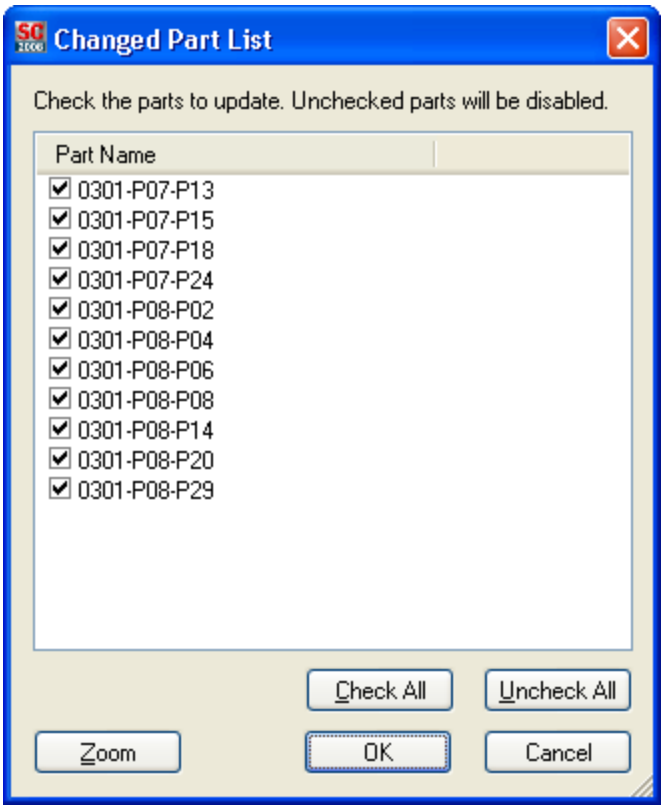
   For more information on the Automatic Nest Settings window, see Automatically Insert and Nest Several Parts (page 266).

Updating Parts in a Nest

After inserting and assigned parts to a nest, they may be modified in their original drawings. To update the parts in the nest to use the latest data, choose SC Plate Nest > Part > Re-insert (page 515). The part will remain in the same location, unless the outer toolpath has changed, in which case, the part will shift slightly. You may need to re-organize the nest after an update, if part geometry has changed.

If the parts have been changed while the nest drawing is open (for example, if another user is modifying the original drawing), a log file will be created during the update. You will need to save and re-open the nest drawing to get the most up-to-date information from the database.

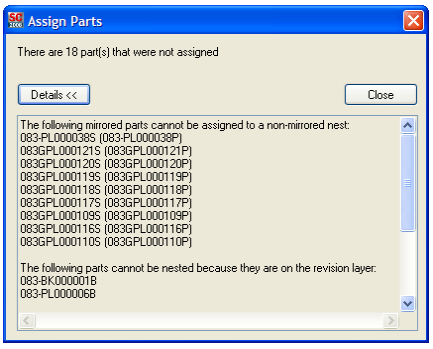
When opening a nest drawing, it will check the drawing for any out-of-date parts. The Changed Part List window is displayed listing those parts, and you may choose which parts to update. Any parts that are not updated are flagged using the Outdated color and moved to the _REV layer. Those parts will need to be manually updated later before being assigned to nests.



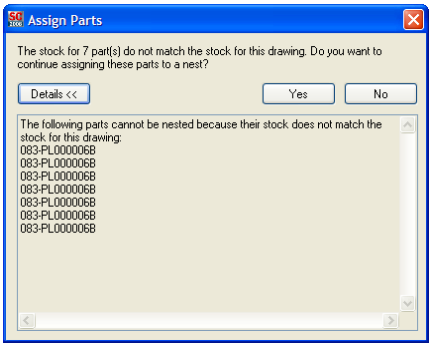
Assign or Unassign Parts to a Nest

Parts placed onto plates are automatically assigned to the nest of that plate. Moving a part off a plate unassigns the part. Parts change colors based on their assigned state. If the part cannot be assigned to the nest, the Assign Parts window will appear informing the user of the problem. Expanding Details will list the reasons and all the parts that cannot be assigned for those reasons.

The most common reasons are: part is on the _REV layer, trying to assign a non-mirrored part to a like/mirror nest (and vice versa), part geometry is invalid (it self-interests, preventing the NC-Machine from getting a complete path), and part is split to another project.



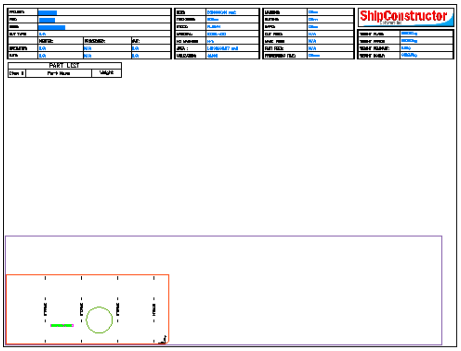
A similar window will also appear when attempting to assign parts of a different stock to the nest.

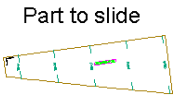

Assign Parts
The stock for 7 part(s) do not match the stock for this drawing. Do you want to continue assigning these parts to a nest?
Details <<
Yes
No
The following parts cannot be nested because their stock does not match the stock for this drawing:
083-PL000006B
083-PL000006B
083-PL000006B
083-PL000006B
083-PL000006B
083-PL000006B
083-PL000006B

Slide a Part Along a Plate Edge or Part Edge

To slide a part along a plate edge or part edge

You should be at a stage where you have a nest and parts in the nest drawing.



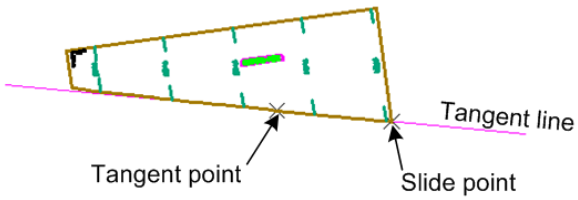


1. Choose SC Plate Nest > Part > Slide Along (page 515)
2. Select the part that you want to move.

   ```
   Select part:
   ```
3. Select the point on the part that you want to follow the cursor.

   ```
   Select slide point on part:
   ```
4. Select a point on the part that will be used in conjunction with the slide point to be the tangent.

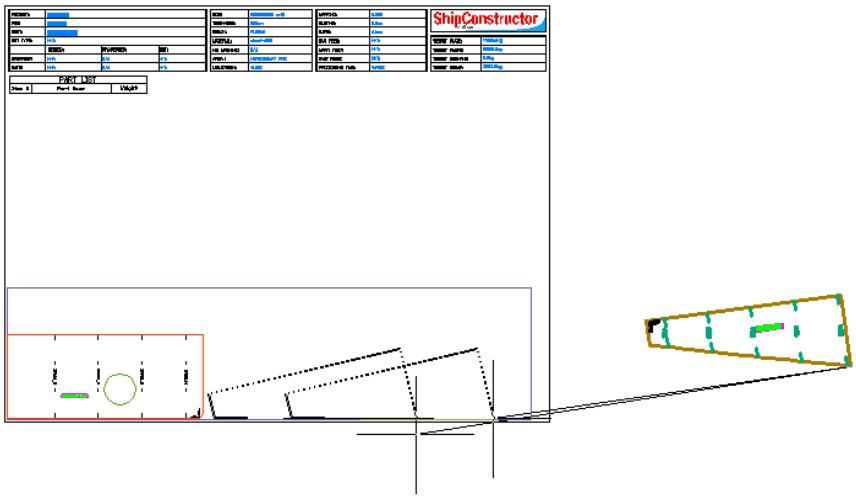
   ```
   Select tangent point on part:
   ```



5. Select the objects that the part will slide along the edge of. This will typically be the plate and nearby parts.

   ```
   Please select stock plate and/or part(s):
   ```
6. You will see an outline of the part you are moving. This indicates the current position of the part.
7. Move the mouse. The part will follow the mouse pointer. The part will rotate so that it will be tangential to the part that it is sliding along. The part automatically keeps the Nest Part Offset distance set in the plate stock library in

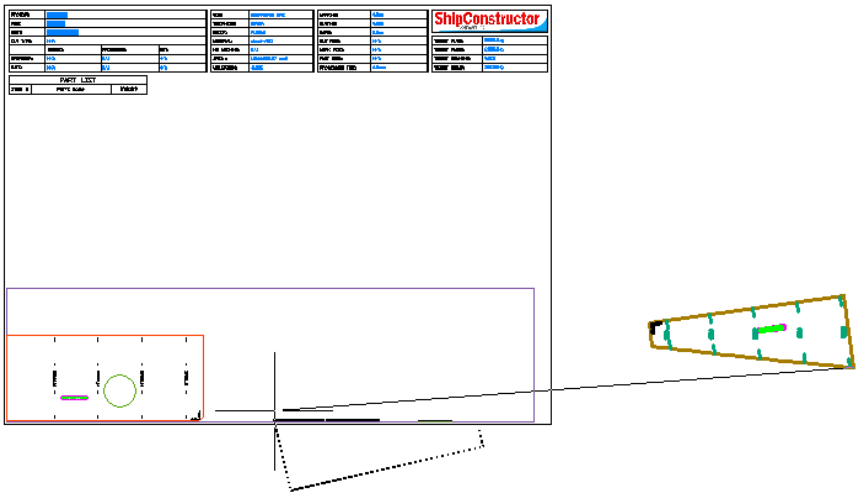
Manager from the nearest part. Make sure that the ghost outline does not intersect with parts or the plate border before final placement of the part.



1. The prompt displays:

```
T - toggle orient mode; R - Rotate 180°:
<Tangent mode>
```

2. If the part is on the wrong side of the part to slide along (sliding inside the other part instead of along the outside), press key R. This toggles the side the part is on.



The slide function supports three slide modes. Press the key T to activate these modes:

- Tangent mode – In this mode the sliding part will be aligned tangentially on the part it is sliding along.
- Two-point mode – The two selected points on the sliding part will be on the outside or inside toolpath of the part it is sliding along. This mode is valuable when placing smaller parts inside holes.
- No rotate mode – The rotation of the part does not change the direction as it slides along.

Practice all three modes. Press the T key to toggle the mode. Move the mouse to see the effect of the three different modes.

3. Make sure you are in the tangent mode. If you are not, press T until the command line displays tangent mode.

Move or Rotate Parts within a Nest

There are three ways you can rotate a part:

- Move or rotate a part
- Rotate a part so it fits within the smallest horizontal or vertical rectangle
- Rotate a part so one side is horizontal or vertical

To move or rotate a part

1. Choose SC Plate Nest > Part > Move/Rotate (page 515)
2. Select the part you are going to rotate or move.

   ```
   Select part:
   ```
3. Select the point which will be the base point for movement and the center of rotation.

   ```
   Specify base point:
   ```
4. The part moves with the mouse.

   You can use the left and right arrow keys to rotate the part. Each press of the function key rotates the part by the angle indicated in brackets on the command line (<5> in our example). Use the plus (+) and minus (-) keys to increase or decrease the increments.

   ```
   Rotate with arrow keys, Modify the step-size with +/- < 5>
   ```

To rotate a part so it fits within the smallest horizontal or vertical rectangle

1. Choose SC Plate Nest > Part > Rotate Shortest Dimension (page 516)
2. Select the part that you want to rotate.
3. Press H to align the part horizontally or V to align the part vertically.

To rotate a part so one side is horizontal or vertical

1. Choose SC Plate Nest > Part > Align with H or V (page 517)
2. Select the part to align.

   ```
   Select part(s):
   ```
3. Select the alignment point by clicking on an edge of the part.

   ```
   Select alignment point:
   ```

   For example, if you want to align the upper edge of the part so it lies horizontal, click the upper edge.

   OSNAP NEAREST and END are automatically turned on.
4. Select the align direction point by clicking any other point on the same edge of the part.

   The two points indicate the direction to align with the vertical or horizontal axis.

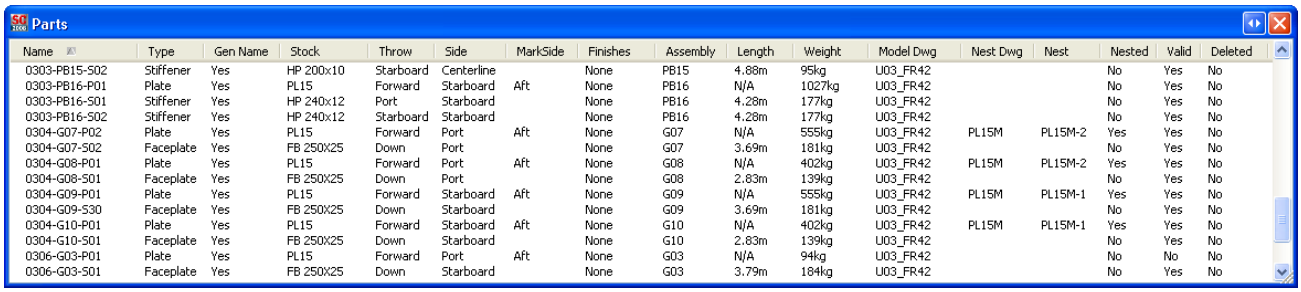
   ORTHO is now on.

   ```
   Select alignment direction point:
   ```
5. Move the mouse until the part is rotated (for example, horizontally).

Finding Parts in a Nest Drawing

To find a part in a nest drawing

1. Choose SC Plate Nest > Part > List (page 517) to open the Parts window.



Name	Type	Gen Name	Stock	Throw	Side	MarkSide	Finishes	Assembly	Length	Weight	Model Dwg	Nest Dwg	Nest	Nested	Valid	Deleted
0303-PB15-S02	Stiffener	Yes	HP 200x10	Starboard	Centerline		None	PB15	4.88m	95kg	U03_FR42			No	Yes	No
0303-PB16-P01	Plate	Yes	PL15	Forward	Starboard	Aft	None	PB16	N/A	1027kg	U03_FR42			No	Yes	No
0303-PB16-S01	Stiffener	Yes	HP 240x12	Port	Starboard		None	PB16	4.28m	177kg	U03_FR42			No	Yes	No
0303-PB16-S02	Stiffener	Yes	HP 240x12	Starboard	Starboard		None	PB16	4.28m	177kg	U03_FR42			No	Yes	No
0304-G07-P02	Plate	Yes	PL15	Forward	Port	Aft	None	G07	N/A	555kg	U03_FR42	PL15M	PL15M-2	Yes	Yes	No
0304-G07-S02	Faceplate	Yes	FB 250X25	Down	Port		None	G07	3.69m	181kg	U03_FR42			No	Yes	No
0304-G08-P01	Plate	Yes	PL15	Forward	Port	Aft	None	G08	N/A	402kg	U03_FR42	PL15M	PL15M-2	Yes	Yes	No
0304-G08-S01	Faceplate	Yes	FB 250X25	Down	Port		None	G08	2.83m	139kg	U03_FR42			No	Yes	No
0304-G09-P01	Plate	Yes	PL15	Forward	Starboard	Aft	None	G09	N/A	555kg	U03_FR42	PL15M	PL15M-1	Yes	Yes	No
0304-G09-S30	Faceplate	Yes	FB 250X25	Down	Starboard		None	G09	3.69m	181kg	U03_FR42			No	Yes	No
0304-G10-P01	Plate	Yes	PL15	Forward	Starboard	Aft	None	G10	N/A	402kg	U03_FR42	PL15M	PL15M-1	Yes	Yes	No
0304-G10-S01	Faceplate	Yes	FB 250X25	Down	Starboard		None	G10	2.83m	139kg	U03_FR42			No	Yes	No
0306-G03-P01	Plate	Yes	PL15	Forward	Port	Aft	None	G03	N/A	94kg	U03_FR42			No	No	No
0306-G03-S01	Faceplate	Yes	FB 250X25	Down	Starboard		None	G03	3.79m	184kg	U03_FR42			No	Yes	No

2. Find the part in the list and double click on it. The drawing zooms to the selected part. The context menu provides the option to zoom to the part in its planar group drawing.

Labelling Parts

To label parts in a nest using the BOM index

1. Choose SC Plate Nest > Auto Label from BOM Index (page 519) to open the Label Properties window.
2. Select the options to use for the labels.
3. Click OK.

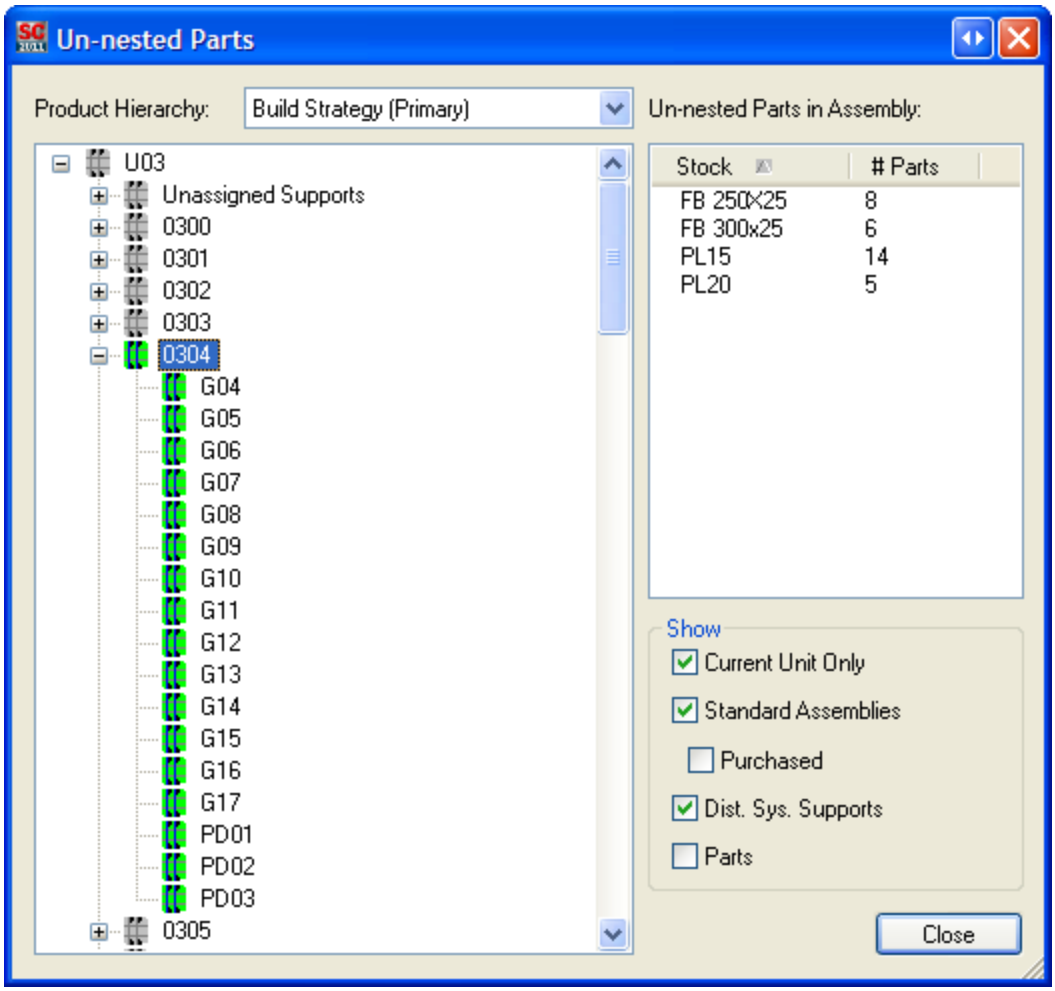
   Labels will be created on each part using the numbering from the BOM index.

To edit the properties of a label

1. Choose SC Plate Nest > Label Properties (page 520) to open the Label Properties window.
2. Modify the settings
3. Click OK.

List Un-nested Parts in an Assembly

1. Choose SC Plate Nest > Part > List Un-nested Parts (page 518) to open the Un-nested Parts window.



2. Select the assembly to list un-nested parts for. If you change the value in the Product Hierarchy dropdownlist, the assembly tree updates to show the assemblies for that product hierarchy.

   Un-nested parts are listed on the right, grouped by stock type.

   Current Unit Only – This option minimizes the amount of information in the tree, speeding up the window, and is especially useful for large projects. This only applies to the primary product hierarchy. For non-primary product hierarchies the entire project is shown.

   Standard Assemblies – If checked, the filter will display assemblies from standard assemblies, and their parts.

   Purchased – If checked, Standard Assemblies tagged as “Purchased” will also be displayed.

   Dist. Sys. Supports – If checked, Distributed Systems Supports are displayed.

   Show Parts – Toggle this option to either display or hide the parts assigned to the assemblies.

Position Parts Next to each other

To position parts next to each other

1. Choose SC Plate Nest > Part > Snap (page 517)
2. Select the parts to move. Press Enter to finish selecting the part. You can also use this function to SNAP several parts at once.

   ```
   Select part(s):
   ```

3. At this point, ShipConstructor activates OSNAP CENTER, INTERSECTION, and END. Select a point on the part you are moving. This point will be base point for the move. Typically you want to select the corner of a part.

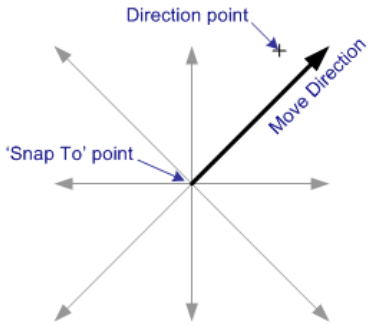
```
Snap point:
```

4. Select the point on the part.

```
Snap to:
```

5. The function expects you to pick the approximate direction that you want to offset the part from the snap point to. The function will snap the closest 45 degrees direction that you pick.

```
Specify direction:
```



6. The part moves by the value of Nest Part Offset or Nest Edge Offset depending on what you selected. This keeps the part the correct distance from the plate edge or other parts.

Hide or Show Nested Parts, Revised Parts, or NC Cutting Paths

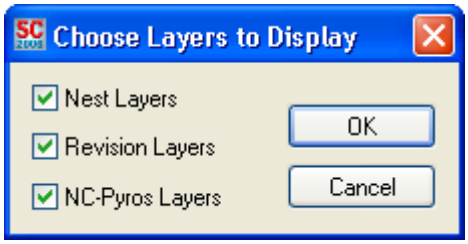
A nest drawing consists of various layers:

- The stock layer contains the stock plate, template border, keywords, and other entities from the template.
- The text layer contains the BOMs.
- The part layer contians parts, either nested or unnested.
- The bridge layer contains all bridges between parts.
- The revision layer contains parts that have been modified since being nested. (When you use SC Plate Nest > Update Nests and BOM (page 522), ShipConstructor automatically moves modified parts to the revision layer.)
- The NC-Pyros layer contains the NC cutting path that you have imported (see Import Cutting Paths into Nests (page 292)).

You can control the display within a nest drawing by hiding or showing each of these layers.

To control the display of nest drawing layers

1. Choose SC Plate Nest > Layers > Visibility (page 518) to open the Choose Layers to Display window.



2. Select the layers you want to display.

   Nest Layers refers to both the stock and part layers.

3. Click OK.

To display all layers in a nest drawing

1. Choose SC Plate Nest > Layers > Activate All (page 519)

To flip between the nest layer and the NC-Pyros layer

1. Choose SC Plate Nest > Layers > Flip NC-Pyros (page 519)

Insert Keywords into a Nest

You can insert keywords into a specific nest (for example, if a specific nest requires a keyword that other nests do not require). If all nests require the same keyword, then insert the keyword into the nest template (see Set Up Nest Templates (page 255)).

To insert keywords into a nest

1. Choose SC Plate Nest > Insert Keywords (page 519) to open the Insert Keyword window.
2. Select the keyword to insert and set the Text Style, Text Size, and Text Rotation.
3. Click OK.
4. Select the location to insert the keyword.

> Note: Make sure you insert the keyword within the nest's outside border. ShipConstructor determines which nest the keyword belongs to based on it being inside those borders.

Edit the Stock Plate of a Nest

To edit the stock plate of a nest

1. Choose SC Plate Nest > Nest > Edit Stock Plate (page 513)
2. Click the nest you want to change.

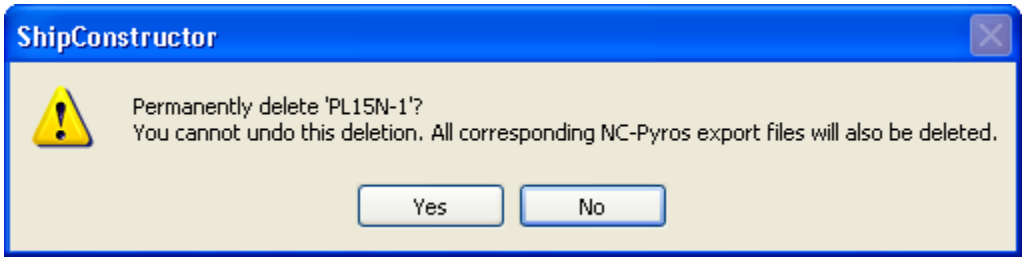
   The Edit Stock Plate window appears.
3. Choose a stock size or change between using a full stock plate or a remnant.

   You cannot change the stock thickness or template drawing.
4. Enter a new name in the Nest Name box.
5. Click OK.

Delete a Nest

To delete a nest from a nest drawing

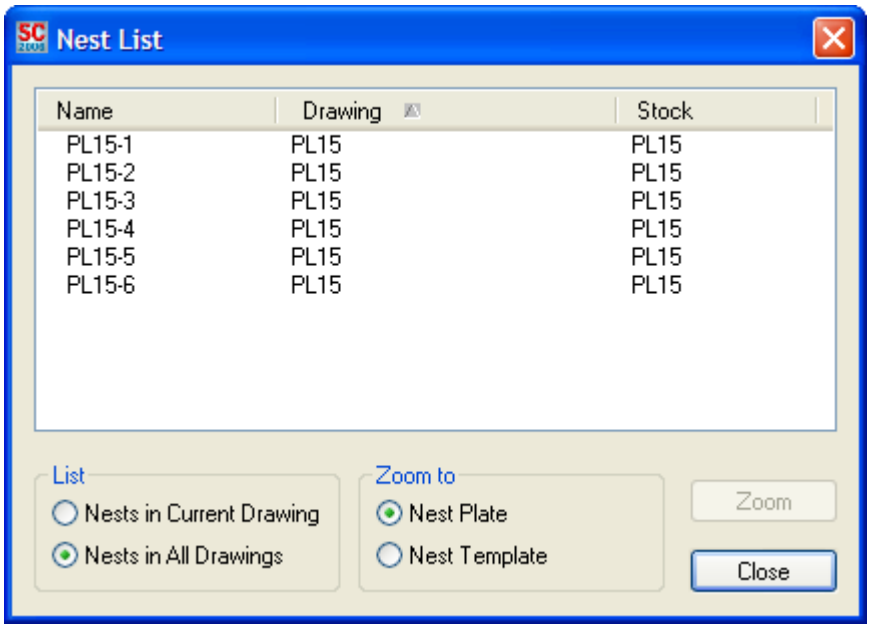
1. Choose SC Plate Nest > Nest > Delete (page 512)
2. Click the nest (either the nest border or the stock plate) to delete.
3. A confirmation dialog will appear because any NC file associated with the nest is also deleted.


ShipConstructor
Permanently delete 'PL15N-1'?
You cannot undo this deletion. All corresponding NC-Pyros export files will also be deleted.
Yes
No

Find a Nest

To find a nest

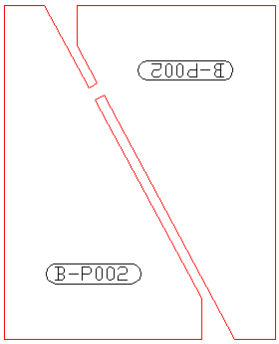
1. Choose SC Plate Nest > Nest > Find (page 512) to open the Nest List window.



2. Select the nest to zoom in to.
3. Click Zoom.

Set Up Bridges

A bridge is a way of connecting two parts together to make cutting, fabrication, and transportation easier. Bridges can be created only between two parts: not between a part and the stock plate, nor between a part and itself (example: across a lightening hole).



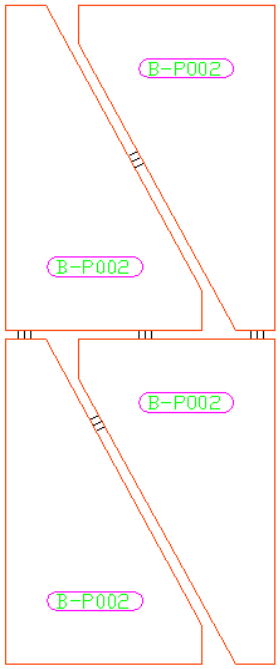
You may want to connect two parts with a bridge for any of the following reasons:

- Reduce piercing – For each part, the cutting machine must pierce a hole through the plate at the start of cutting. This takes time, especially for thick plates, and can wear out the cutting tool. By connecting parts with a bridge, you can reduce the number of holes the cutting machines must pierce through the plate.
- Lost parts – Small parts can fall through the grid that the plate rests on during cutting and are easily lost.
- Minimize warping – Small, slender parts can warp during cutting. Bridges help them maintain their shape.
- Keep parts together – You can use bridges to keep all parts together to make transportation easier.

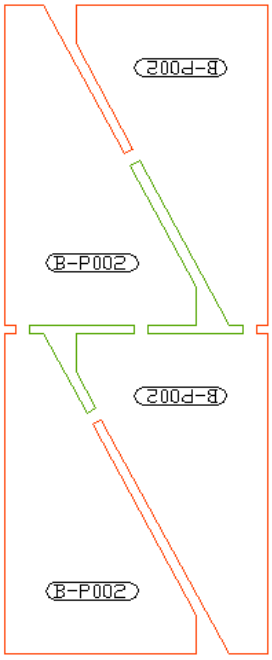
When you insert bridges into a nest, ShipConstructor keeps track of the bridges separately from the nest and parts. That is, ShipConstructor does not modify the nest or the parts. This allows you to update parts in a nest (for example, if a piping penetration has been added to a part) even after you have inserted bridges, without having to re-insert the bridges.

However, if you update the outside of a part, ShipConstructor may not be able to maintain the existing bridge. In this case, you have to re-insert any bridge that is not maintained.

When you export a nest to NC-Pyros, ShipConstructor integrates the part outlines and bridges into a single outline.

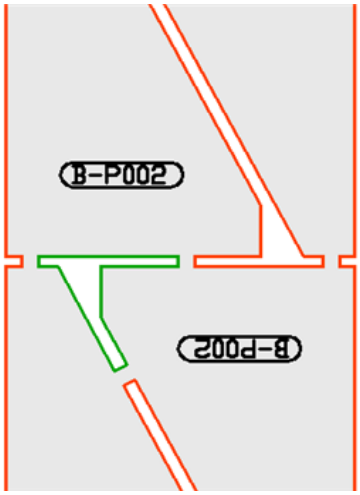


Parts with bridges as they appear within the nest in ShipConstructor



A bridge as it appears when the nest is exported to NC-Pyros

Usually you connect two parts with only one bridge. However, there may be cases where you need to insert two or more bridges between two parts. This results in toolpaths on the outside of a part that ShipConstructor treats as if they were holes. In the following example, three parts are connected by three bridges. ShipConstructor colors the resulting hole green.



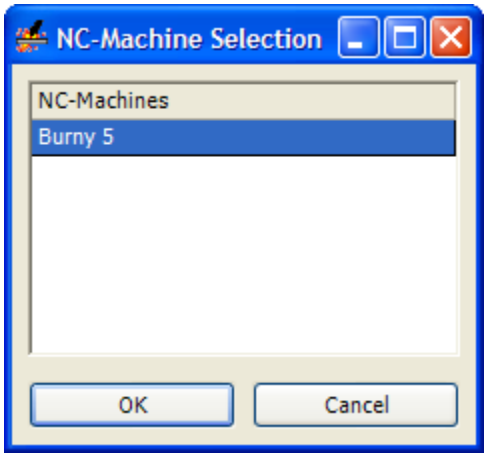
Assign to NC-Machine

You must first assign an NC machine to a nest before inserting bridges. The width of each bridge is defined when assigning the NC machine to plate stock.

To assign a nest to an NC-Machine

1. Choose SC Plate Nest > Nest > Assign to NC-Machine (page 513)
2. Click the nest you want to assign.

   The NC-Machine Selection window appears.



3. Select the NC-Machine.
4. Click OK.

Insert a Bridge

To insert a bridge

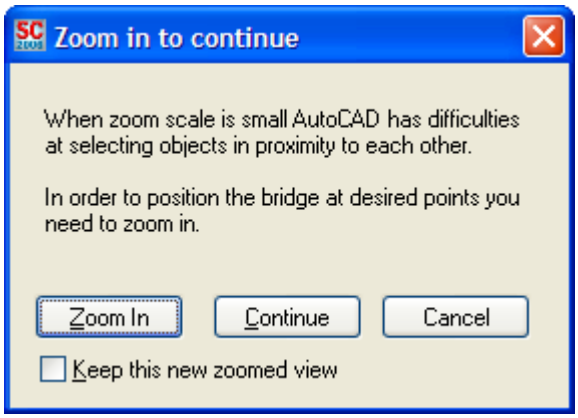
1. Choose SC Plate Nest > Bridge > New (page 525)

   ShipConstructor sets the OSNAP to NEAREST.
2. Select a position on a part for the start of the bridge.

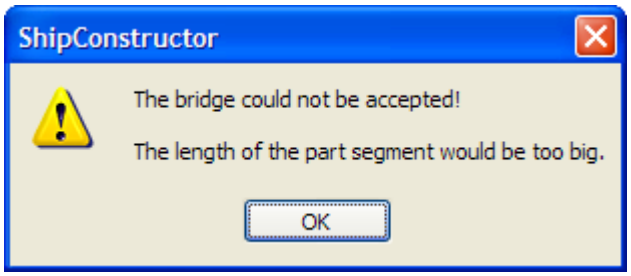
   ShipConstructor sets the OSNAP to PERPENDICULAR.
3. Select a position on the other part for the end of the bridge.

ShipConstructor inserts the bridge.

Under certain conditions, the following error message may appear.


Zoom in to continue
When zoom scale is small AutoCAD has difficulties at selecting objects in proximity to each other.
In order to position the bridge at desired points you need to zoom in.
Zoom In
Continue
Cancel
Keep this new zoomed view

Select ZoomIn and try again.



If a message appears telling you that the bridge cannot be accepted, the proposed bridge violates a set condition (for example, one end of the bridge may be too close to a sharp corner of one of the parts).

Delete Bridges

To delete bridges

1. Choose SC Plate Nest > Bridge > Delete (page 525)
2. Select the bridges that you want to delete, or type All to delete all bridges in the nest drawing.
3. Press Enter.

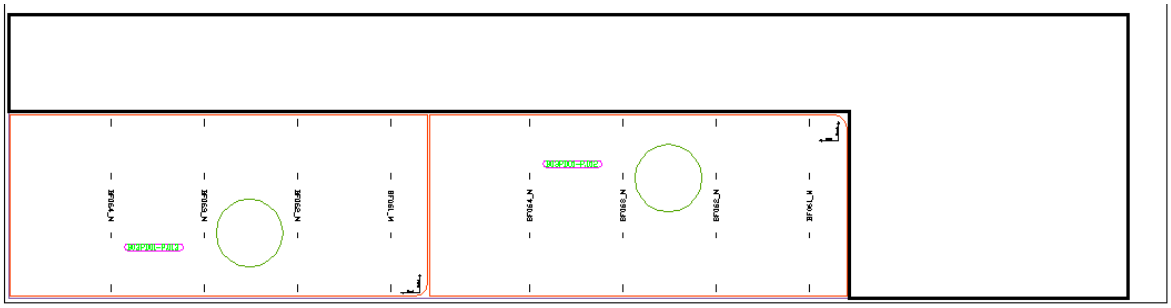
Set Up Remnants

Remnants are unused portions of a nest plate. In most cases you can use remnants later in your project for a new nest.

Create a Remnant

To create a remnant

1. Draw a closed polyline representing the remnant outline and close the last leg of the polyline.

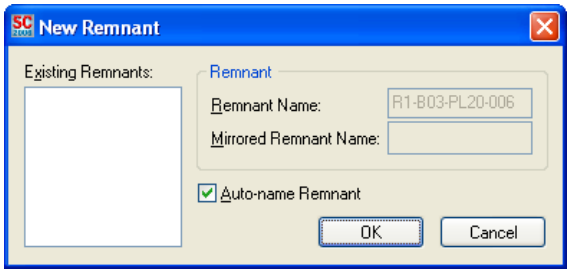


2. Choose SC Plate Nest > Remnant > New (page 513)
3. Click on the remnant outline (polyline).

```
Select Polyline:
```

Next, you need to tell ShipConstructor which portion of the remnant will be used to cut off the remnant from the nest plate. You do this by defining a cut line.

4. The New Remnant window appears.

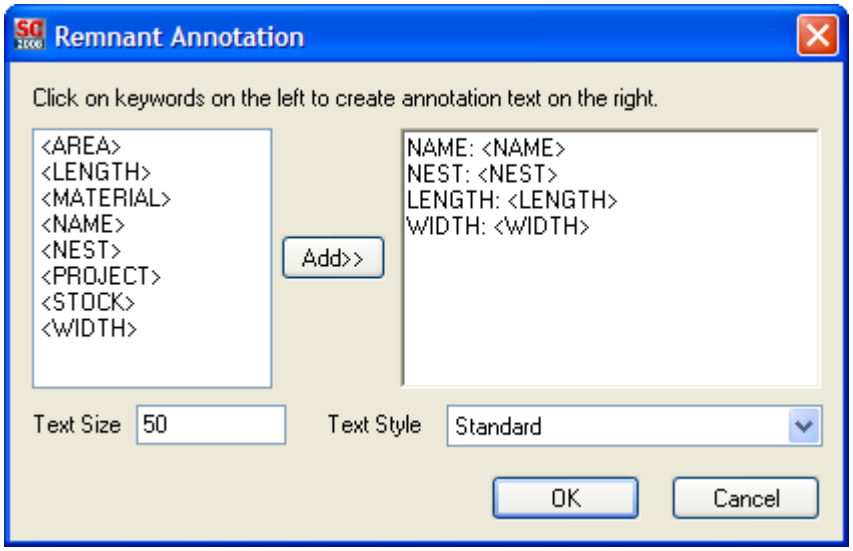


Enter a Remnant name.

When the plate is a like and mirror cut, you must enter two names: one for the remnant and one for the mirrored remnant. You can later use the two remnants individually for like nests only.

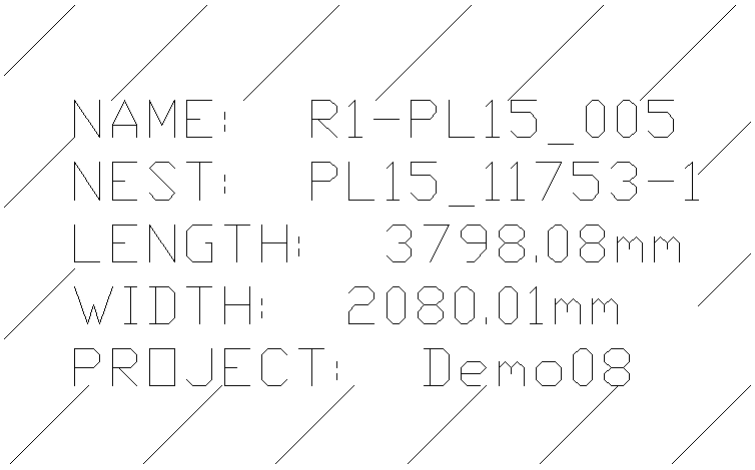
5. Click OK.

The Remnant Annotation window appears.



6. Select the keywords to use for the remnant marking and set the Text Size (in model space units).
7. Click OK.

The remnant becomes hatched.



You can now use the remnant for a new nest.

Edit a Remnant

After a remnant is defined, its shape and label can be modified, even if the remnant is being used by a nest, but as long as that nest has not been issued or cut.

To edit a remnant's shape

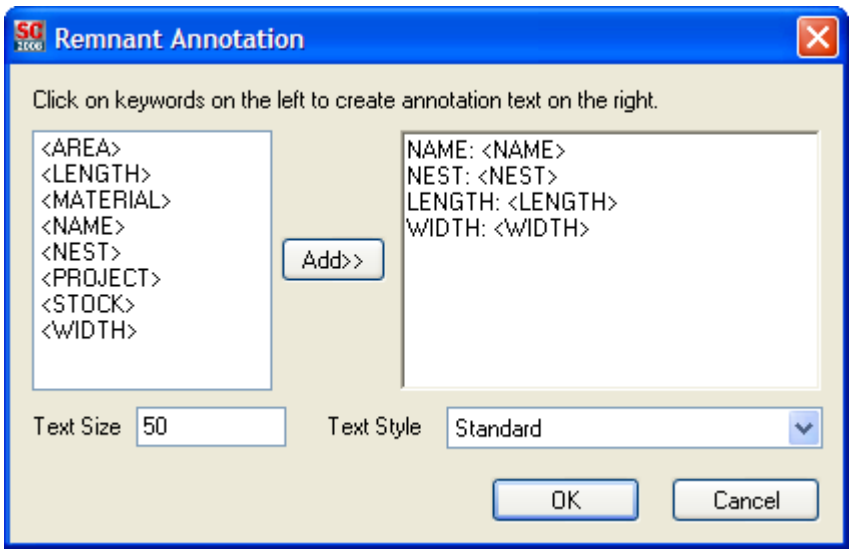
1. Choose SC Plate Nest > Remnant > Edit Remnant (page 514)
2. Select the remnant to modify.
3. Click on the new remnant outline (polyline).

```
Select Polyline:
```

   The nest using this remnant will be automatically updated with its new shape.

To edit a remnant's label

1. Choose SC Plate Nest > Remnant > Edit Remnant Label (page 514)
2. Select the remnant to modify.
3. The Remnant Annotation window appears.



4. Modify the keywords used by the remnant's label.

5. Click OK.

Delete a Remnant

Note: You can also use the AutoCAD **ERASE** command to delete a remnant.

To delete a remnant

1. Choose SC Plate Nest > Remnant > Delete (page 514)
2. Select the remnant to delete.

Note: You cannot delete a remnant that is already used as a nest.

Check and Fix Nests

Before you issue any nests for processing, you must check them. It is vital to check your nests to detect and repair any possible problems.

Check Nest Drawings for Errors and Un-nested Parts

The checks performed are run on each nest in the drawing. A complete list is as follows: all parts are correctly assigned to the nest, all remnants created from the nest use the correct geometry, all bridges in the nest are properly connected to parts, all assigned parts are up-to-date, and the part geometry is valid. When the checks are expanded to the entire drawing, additional checks are run on all unassigned parts to confirm they are in the correct drawing.

To check nest drawings for errors

1. Choose SC Plate Nest > Check Nests and BOM (page 521)

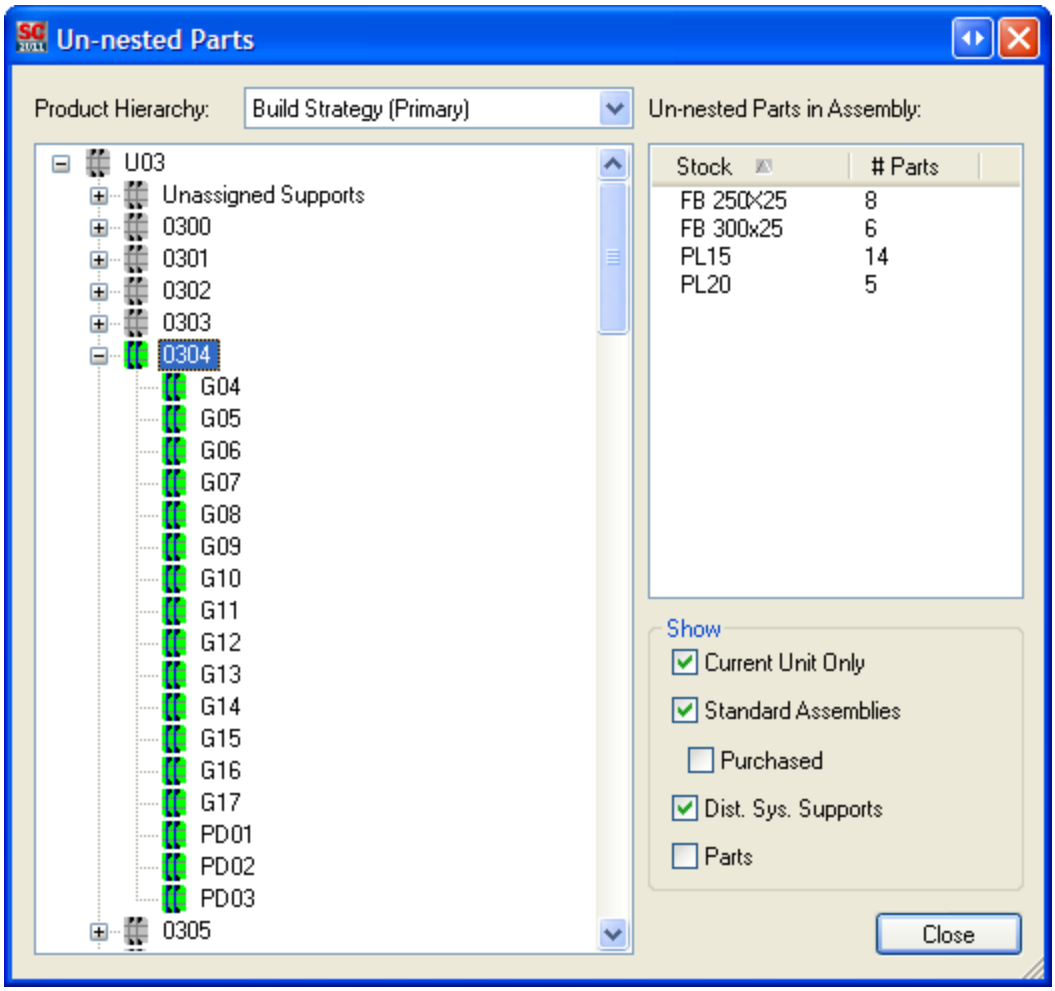
   ShipConstructor checks all nests, reports any errors, updates all BOMs, and lists all un-nested parts.

```
7/10/2007 9:52:21 AM     Log file 'D:\Projects2008\Demo08\LogFiles\PlateNests\Check PL15 2007-07-10 09.52.21.txt' created by shipconstructor

Checking nests in drawing 'PL15.DWG'
Checking each nest for:
* all parts are correctly assigned to the nest
* all remnants created from the nest use correct geometry
* all bridges in the nest are properly attached to parts
* all assigned parts are up-to-date
        Checking parts in drawing
        Checking parts in database
                Warning: '0303-07-S02' has a different stock thickness than: 'PL15.DWG'
                Warning: '0303-07-C01' has a different stock than: 'PL15.DWG'
        Checking nests in database
        Checking nests in drawing
                Checking nest 'PL15-1'
                Done checking nest 'PL15-1'
                Checking nest 'PL15-2'
                Done checking nest 'PL15-2'
                Checking nest 'PL15-3'
                Done checking nest 'PL15-3'
                Checking nest 'PL15-4'
                        Error: '0303-06-P03' not assigned to: 'PL15-4'; Mismatched Like/Mirror
                        Warning: '0303-06-P01' is out-of-date
                        Warning: '0303-06-S01' is out-of-date
                        Warning: '0303-06-S02' is out-of-date
                        Warning: '0303-06-C01' is out-of-date
                Done checking nest 'PL15-4'
                Checking nest 'PL15-5'
                Done checking nest 'PL15-5'

Check Complete
1 Error(s), 0 Fixed, 6 Warning(s)
```

To check nest drawings for un-nested parts

1. Choose SC Plate Nest > Part > List Un-nested Parts (page 518)

   ShipConstructor lists all un-nested parts.



Update Nest Keywords and BOMs

To update nest keywords and BOMs

1. Choose SC Plate Nest > Update Nests and BOM (page 522)

Check Nests for Collisions

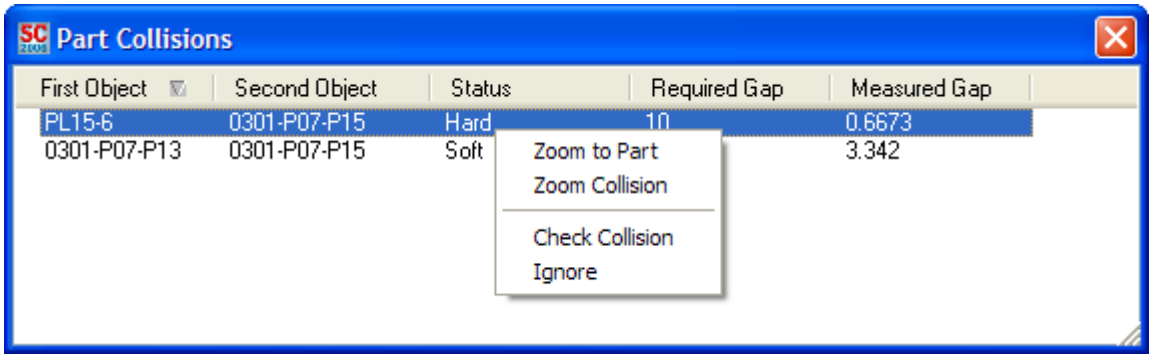
If you manually nest or re-insert parts into a nest, you should check the nest for collisions. ShipConstructor can tell you if any parts, remnants, or nests are too close together or overlapping.

To check nests for collisions

1. Choose SC Plate Nest > Nest Collision Check (page 522)

   It may take about one minute to check the nests.

   The Part Collisions window appears.



Each pair of problem parts is listed with the type of problem under Status:

- Hard – The boundaries of the two parts intersect.
- Soft – The two parts do not intersect, but they are closer together than the Plate Gap set for this plate stock in Manager. See Plate Stock Properties (page 52)
- Inside – One of these parts is completely inside the other part.
- Fixed – The detected problem has been fixed.
- Ignored – You specify that you want to ignore this problem.

When you select a collision in the list, ShipConstructor will highlight the two parts in the drawing and draw circles around the intersection points of the collision.

2. Right-click an item in the list to do the following:
   - Zoom to Part – Zooms to the part.
   - Zoom to Collision – Zooms to the collision of the two parts.
   - Check Collision – Checks the currently selected collision to see if the problems have been fixed. (If you have moved these parts around, ShipConstructor will detect new problems with other parts and add them to the bottom of the list.)
   - Ignore – Tells ShipConstructor to ignore this problem.
3. Correct the collisions and re-run the nest check.

Re-insert Parts Into a Nest

After you check your nest drawings (see Check Nest Drawings for Errors and Un-nested Parts (page 288)), ShipConstructor isolates all parts that require re-nesting.

After re-inserting parts, make sure you check the nests for collisions (see Check Nests for Collisions (page 289)).

To re-insert parts into a nest

1. Choose SC Plate Nest > Part > Re-insert (page 515)
2. Select the parts to re-insert. The parts are updated to their current version that is in the database. If any parts being re-inserted have been changed since the nest drawing was last opened, a log file will be displayed listing which parts are out-of-date. The nest drawing will need to be saved and re-opened to get the latest part information.

Generate NC Code for Nests

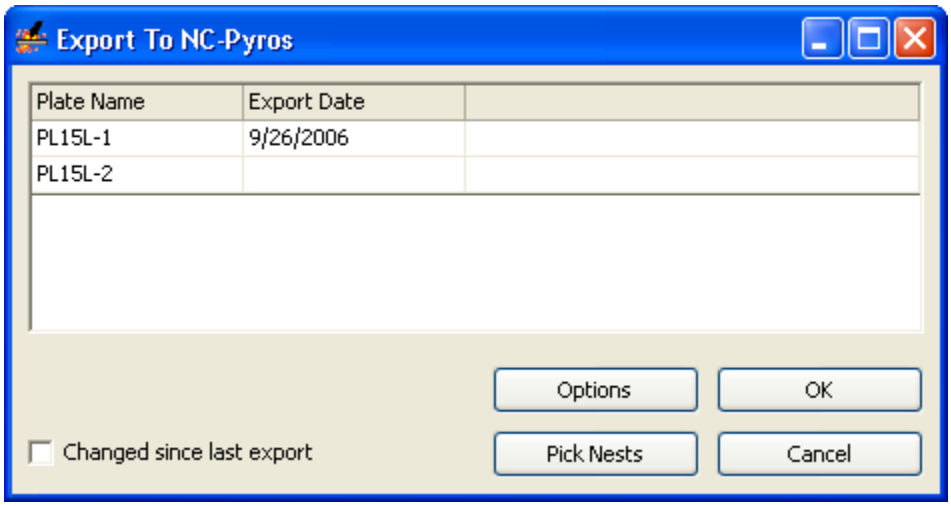
Export Nests to NC-Pyros

You can export nests to a DXF file that you can then read into NC-Pyros or other NC code generation programs for creating cutting paths. A check is run on the nests being exported to confirm their parts are up-to-date.

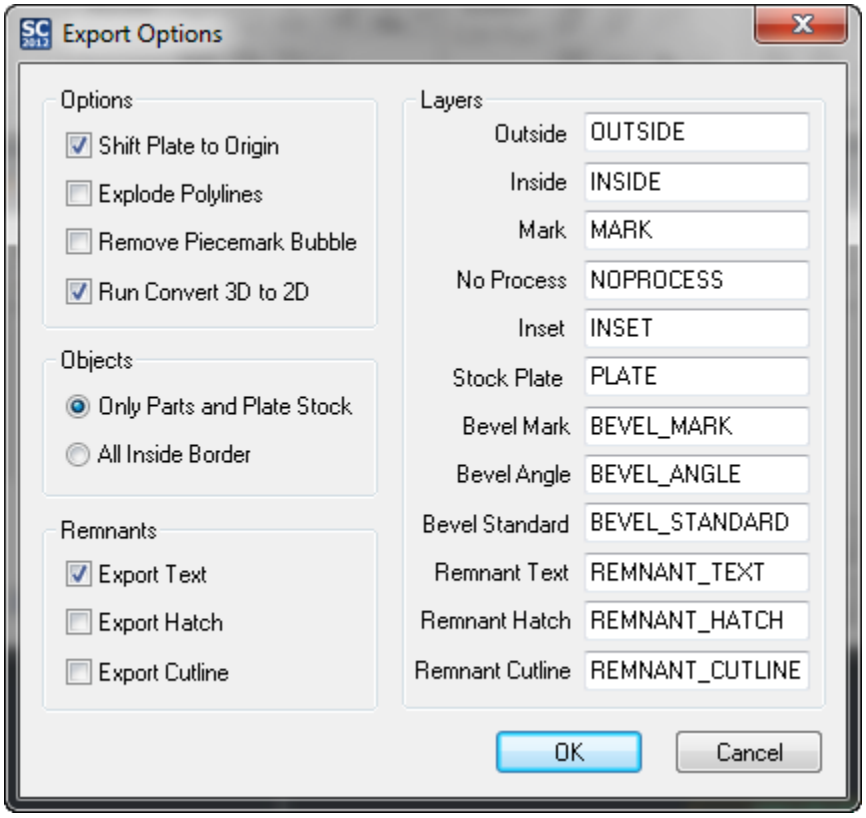
If any of the plate parts exported have bevels, then there will be “0” bevel text objects on segments adjacent to the bevels added to the DXF file. NC-Pyros uses these “0” texts for calculating bevel locations.

To export nests to NC-Pyros

1. Choose SC Plate Nest > NC-Pyros > Export To (page 523) to open the Export To NC-Pyros window.

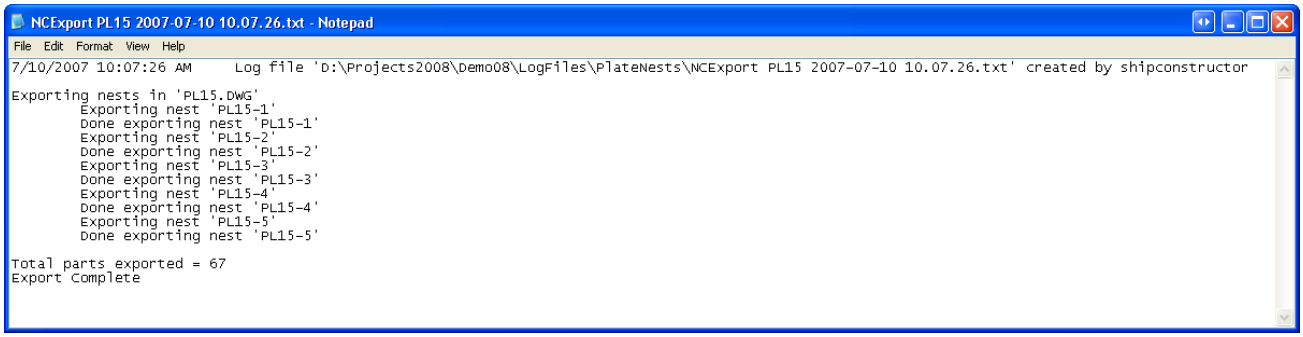


2. Select the nests to export.
3. Click Options and set the options.



4. Click OK.
5. Click OK.

   A log file displays the results of the export.


NCExport PL15 2007-07-10 10.07.26.txt - Notepad
File Edit Format View Help
7/10/2007 10:07:26 AM Log file 'D:\Projects2008\Demo08\LogFiles\PlateNests\NCExport PL15 2007-07-10 10.07.26.txt' created by shipconstructor
Exporting nests in 'PL15.DWG'
Exporting nest 'PL15-1'
Done exporting nest 'PL15-1'
Exporting nest 'PL15-2'
Done exporting nest 'PL15-2'
Exporting nest 'PL15-3'
Done exporting nest 'PL15-3'
Exporting nest 'PL15-4'
Done exporting nest 'PL15-4'
Exporting nest 'PL15-5'
Done exporting nest 'PL15-5'
Total parts exported = 67
Export Complete

Generate NC Code for Nests

You can use NC-Pyros to generate NC code for your nests. After generating cutting paths, you should export the cutting paths as a DXO file so you can import them into ShipConstructor (see Import Cutting Paths Into Nests (page 292)). See the NC-Pyros manual for details.

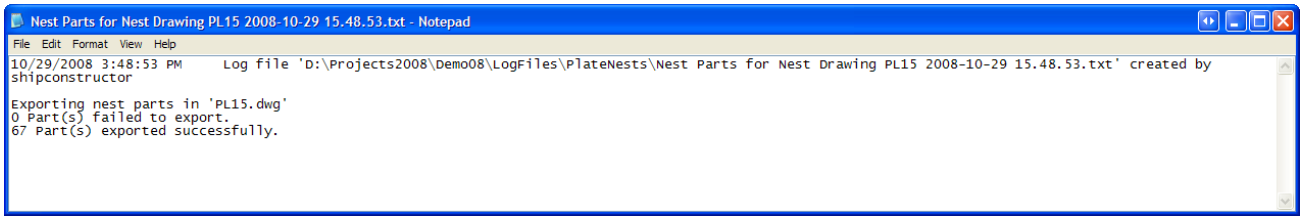
Export Parts to Individual Drawings

Some user may use nesting software other than NC-Pyros. The Export Nest Parts commands allow the user to create a new drawing file, in either DWG or DXF format, for each of the parts in the current nest drawing, using the part's name as the filename. Duplicate part names will be appended with a "_1", "_2", etc. Like/Mirror parts use the Like part's name.

The export will also use the Nest Export options as set in the Options window in Step 3 from Export Nests to NC-Pyros (page 290)

To export parts to individual drawings

1. Choose SC Plate Nest > Part > Export Nest Parts to DWG (page 518) or SC Plate Nest > Part > Export Nest Parts to DXF (page 518). Select the parts to select (or all).
2. Select the folder location to create the part drawings in.
3. A log file displays the results of the export.


Nest Parts for Nest Drawing PL15 2008-10-29 15.48.53.txt - Notepad
File Edit Format View Help
10/29/2008 3:48:53 PM Log file 'D:\Projects2008\Demo08\LogFiles\PlateNests\Nest Parts for Nest Drawing PL15 2008-10-29 15.48.53.txt' created by shipconstructor
Exporting nest parts in 'PL15.dwg'
0 Part(s) failed to export.
67 Part(s) exported successfully.

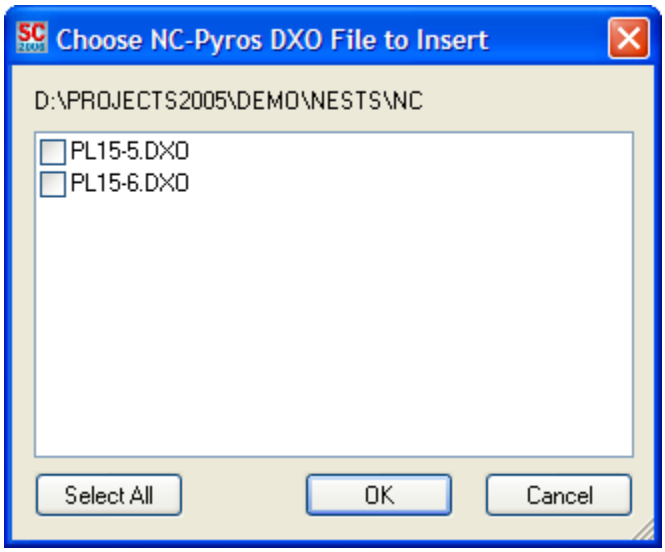
Import Cutting Paths Into Nests

After generating cutting paths for your nests in NC-Pyros, you should import the cutting paths back into ShipConstructor so you can overlay the nested parts with the actual cutting path (see Hide or Show Nested Parts, Revised Parts, or NC Cutting Paths (page 280)). When you print the nests, you can also print the cutting paths so that you know the path the machine will take.

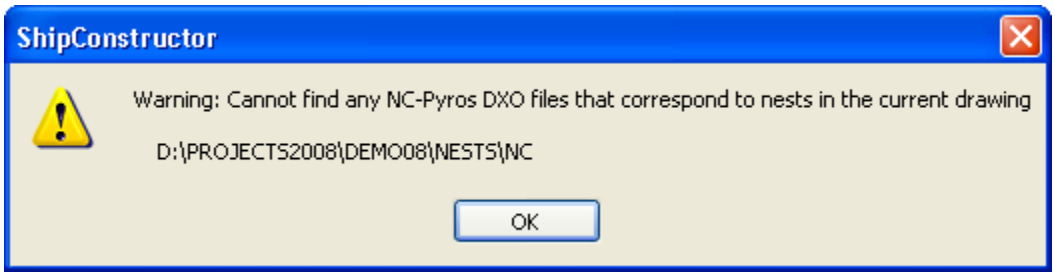
When you import cutting paths into nests, a time estimate is also imported. After importing cutting paths into nests, and before printing nests, you should update your nests and BOMs so that this time estimate is included in the nests. See Update Nest Keywords and BOMs (page 289).

To import cutting paths into nests

1. Choose SC Plate Nest > NC-Pyros > Import From (page 525) to open the Choose NC-Pyros DXO File to Insert window.



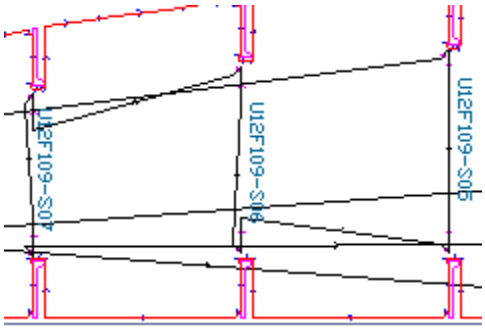
Note: When exporting the DXO file from NC-Pyros, you must save the file to the appropriate folder of the ShipConstructor project. Otherwise, an error message appears when you try to import cutting paths into ShipConstructor.


ShipConstructor
Warning: Cannot find any NC-Pyros DXO files that correspond to nests in the current drawing
D:\PROJECTS2008\DEMO08\NESTS\NC
OK

2. Select the DXO file (exported from NC-Pyros) for the nest.
3. Click OK.

   It may take some time to import the cutting paths.

   ShipConstructor overlays the cutting paths over the nests.



Update Nest BOMs

After importing cutting paths into nests and before printing nests, you should update your nests and BOMs so that NC-Pyros data is included in the header. See Update Nest Keywords and BOMs (page 289).

Plot Nests

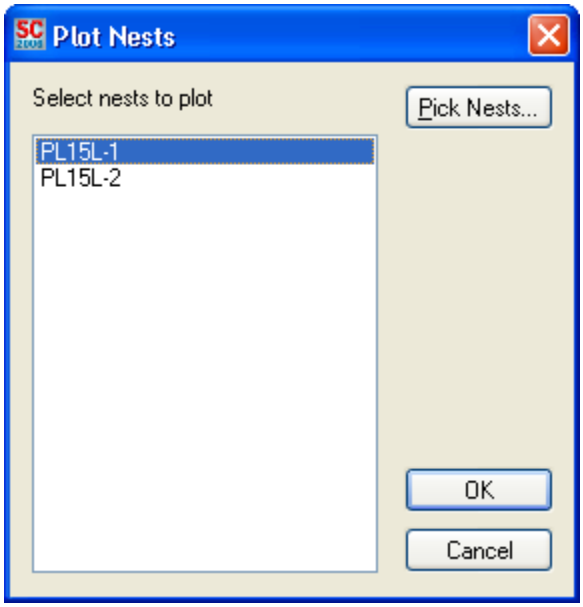
You usually plot each nest on a small sheet of paper (for example, 11” x 17” or A3 paper) and then combine the printed sheets with other drawings in a work package.

Before plotting multiple nests, set your default printer to the printer you want to plot your nests to and set the default paper size and orientation for this printer. You should also plot a single nest using AutoCAD’s **PLOT** command to check all settings.

When plotting multiple nests, ShipConstructor automatically sets the plotted area to the outside border of the nest and plots each nest on a separate plot.

To plot nests

Choose SC Plate Nest > Nest > Plot (page 513) to open the Plot Nests window.



1. Select the nests to plot by doing either of the following:
   - Click the nests in the list.
   - Click Pick Nests, click the nests in the drawing, and press Enter.
2. Click OK.

Generate Nest Reports

See the Project Management manual for information on generating reports.

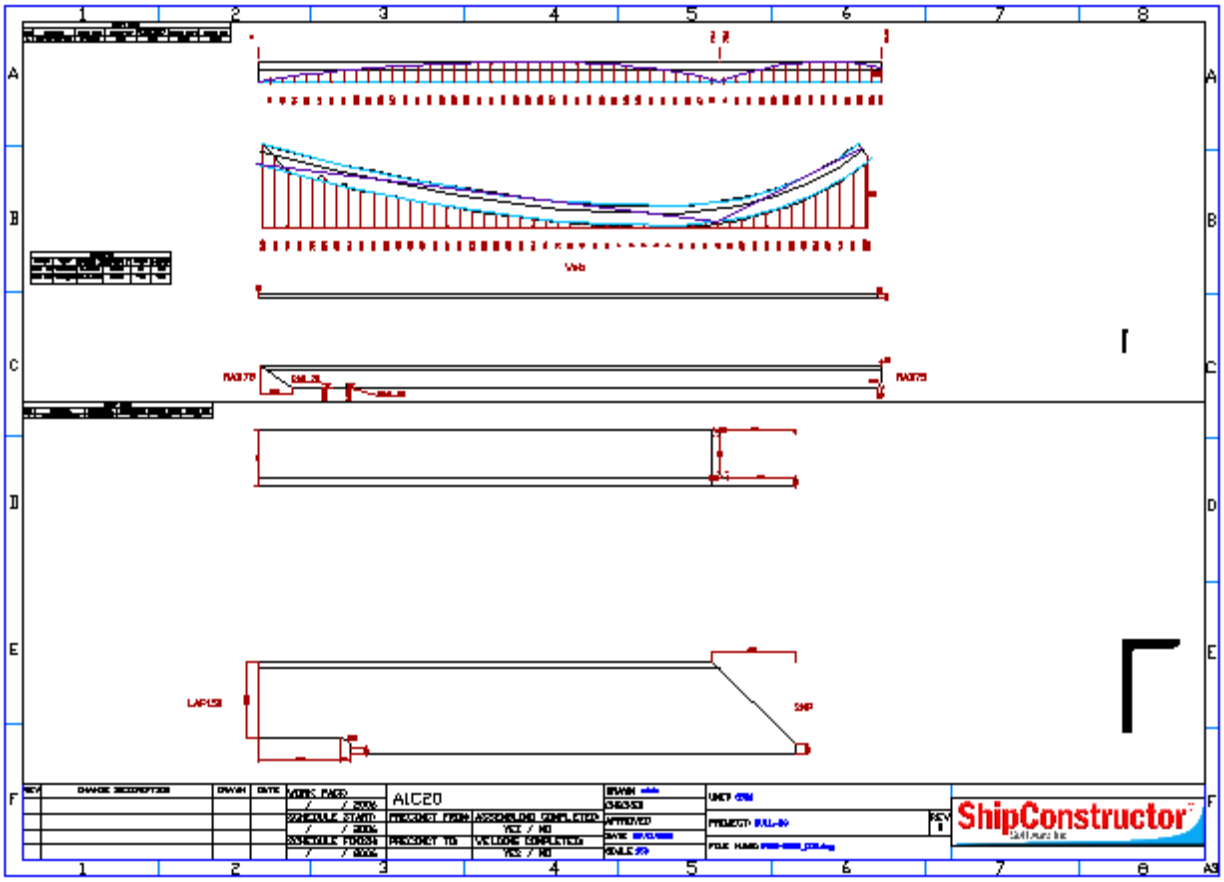
Prepare Work Package

A work package is a way of assembling all of the information that production needs into a single package. For example, you can assemble a package containing nest reports, plots, and NC tapes.

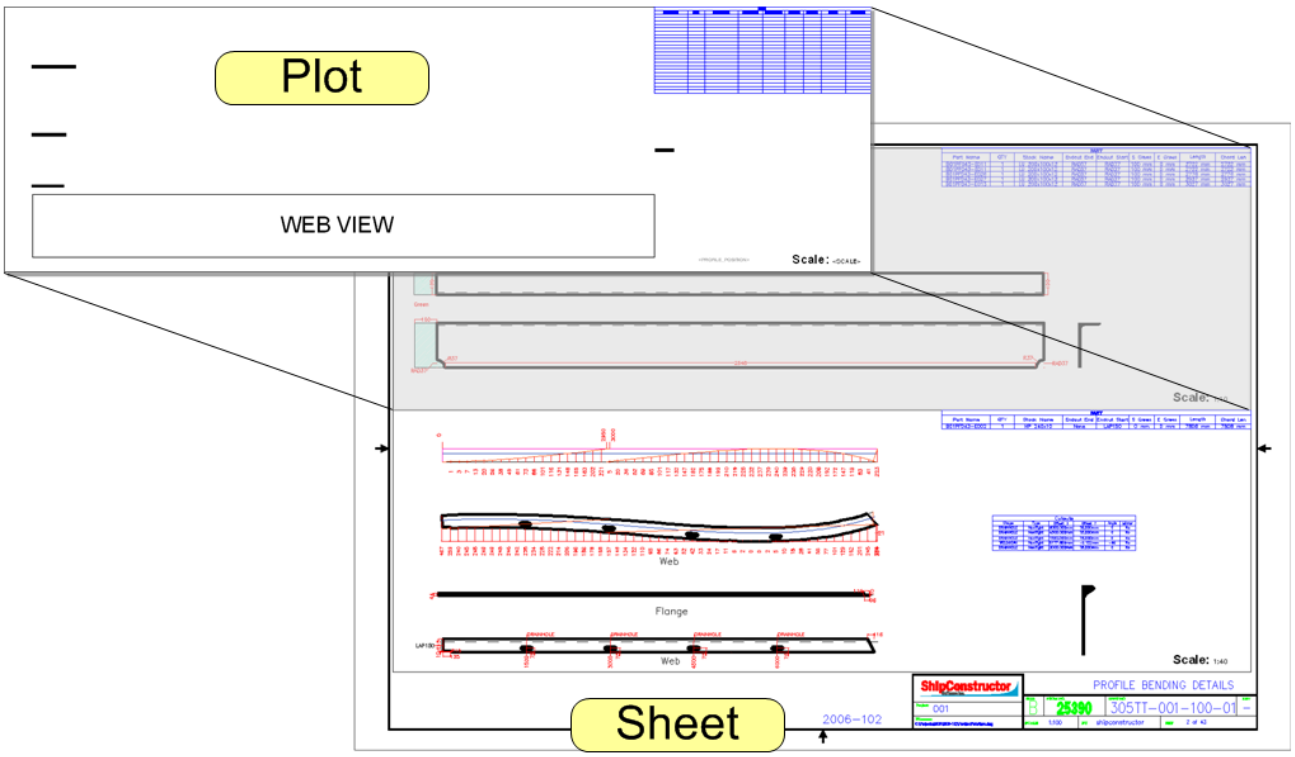
Profile Plots

A profile plot (also known as a stiffener plot) is a detailed drawing of a stiffener and contains the information that production needs to fabricate that stiffener. A profile plot consists of three views of the stiffener: flange, web, and profile. ShipConstructor groups stiffeners with the same stock and endcuts into one plot.

A profile plot can contain inverse bending information for curved stiffeners.



When you create a profile plot drawing, ShipConstructor creates a copy of the drawing template. When you insert profile plots into a profile plot drawing, ShipConstructor uses the plot template for the layout.



Set Up Profile Plot Templates

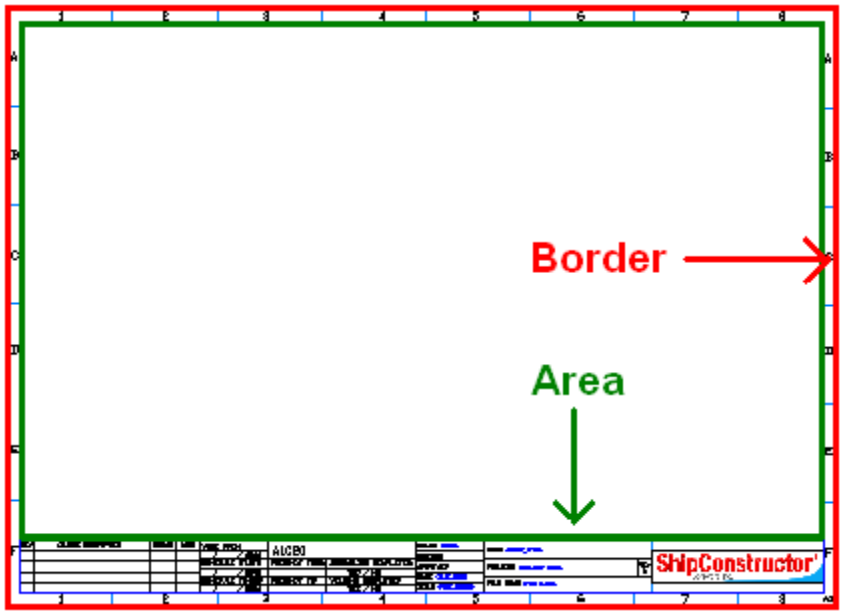
Profile plot templates control the layout of profile plots. Profile plots use three types of profile plot templates.

Profile Plot Drawing Template

The profile plot drawing template should contain standards for text, dimensions, layers and line types. There should be no graphical entities in this template.

Profile Sheet Template

The profile sheet template represents a piece of paper. It contains the title block, border, and an area where individual plots are placed inside.



An example plot sheet template

Create a Profile Sheet Template

To create a profile sheet template

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.
2. Select the Templates page.
3. Select the ProfilePlotSheet folder.
4. Click New ProfilePlotSheet.

Define the Drawing Border of a Profile Plot Sheet Template

A plot template must contain a border and an area. The border defines the outer extents of the sheet, and is used to calculate the tiling of multiple sheets in a profile plot drawing. The area defines the rectangular area in which profile plots will be placed. If large enough, multiple profile plots can be tiled within this area.

To draw a border for the profile sheet

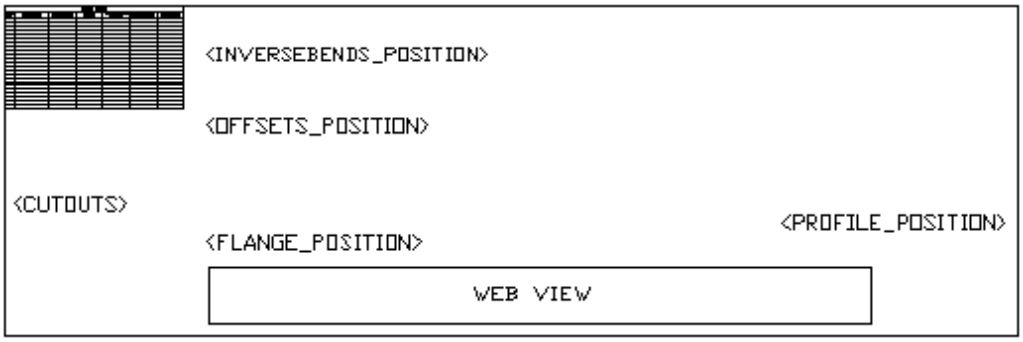
1. In a profile sheet template drawing, choose SC Profile Sheet Template > Draw Border (page 543)
2. Select two points to define the diagonal corners of the rectangular border.

To draw an area for the profile sheet

1. In a profile sheet template drawing, choose SC Profile Sheet Template > Draw Area (page 543)
2. Select two points to define the diagonal corners of the rectangular area. The selected points should be inside the border.

Profile Plot Template

The profile plot template contains the template layout of how individual stiffeners will be detailed. This is where you add keywords for the different views, the cutout information table and any other plot specific information. This is also typically where a BOM would be placed.



An example plot template

Create a Profile Plot Template

To create a profile plot template

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.
2. Select the Templates page.
3. Select the ProfilePlot folder.
4. Click New ProfilePlot.

Define the Drawing Border of a Plot Template

A plot template must contain a border. The border defines the outer boundary of a plot and must fit within the area defined in any profile sheet templates this template will be used with.

To define the drawing border of a plot template

1. In a profile plot template drawing, choose SC Profile Plot Template > Draw Border (page 539)
2. Select two points to define the diagonal corners of the rectangular border.

Define the Web View Area of a Plot Template

A plot template must contain a web view area. The web view area defines the area where the view of the stiffener web will be placed. The horizontal width and vertical height of the web view area are used to scale the stiffener in this and subsequent views within this profile plot.

To define the web view area of a plot template

1. Draw a rectangle (within the plot template drawing border) defining the area for profile plots.
2. Choose SC Profile Plot Template > Draw Web View Area (page 540)
3. Select the rectangle that you drew in Step 1.

Insert Keywords into a Profile Plot Template

You can insert keywords into plot sheet templates and plot templates.

To insert keywords into a profile plot template

1. Choose SC Profile Plot Template > Insert Keywords (page 540) to open the Insert Keyword window.

   Note: The list of keywords is different for plot sheet template and plot templates.

2. Select the keyword you want to insert.
3. Set the Text Size and Text Rotation.
4. Click OK.
5. Select the insertion point for the keyword.

Insert a BOM Table into a Profile Plot Template

You can now predefine a BOM definition for use within a profile plot template. This enables you to format an empty BOM table to your specifications (correct text style, size, and so on).

Note: To insert the BOM table, there must already be a profile plot BOM definition defined. See BOM Definitions Manager (page 47).

To insert a BOM table into the profile plot template

1. Choose SC Profile Plot Template > Insert BOM Table (page 542) to open the BOM Definition Wizard window
2. Select a Profile Plot BOM.
3. Click Next.
4. Modify the desired Table and label options.
5. Click Finish.
6. Select the location to place the empty BOM.

Updating ShipConstructor2005 Keywords

To help update ShipConstructor2005 templates so they can be used in ShipConstructor there is a command to replace any known ShipConstructor2005 keywords in a template with the corresponding ShipConstructor keywords.

To replace legacy keywords in a profile plot template

1. Choose SC Profile Plot Template > Replace Legacy Keywords (page 543)

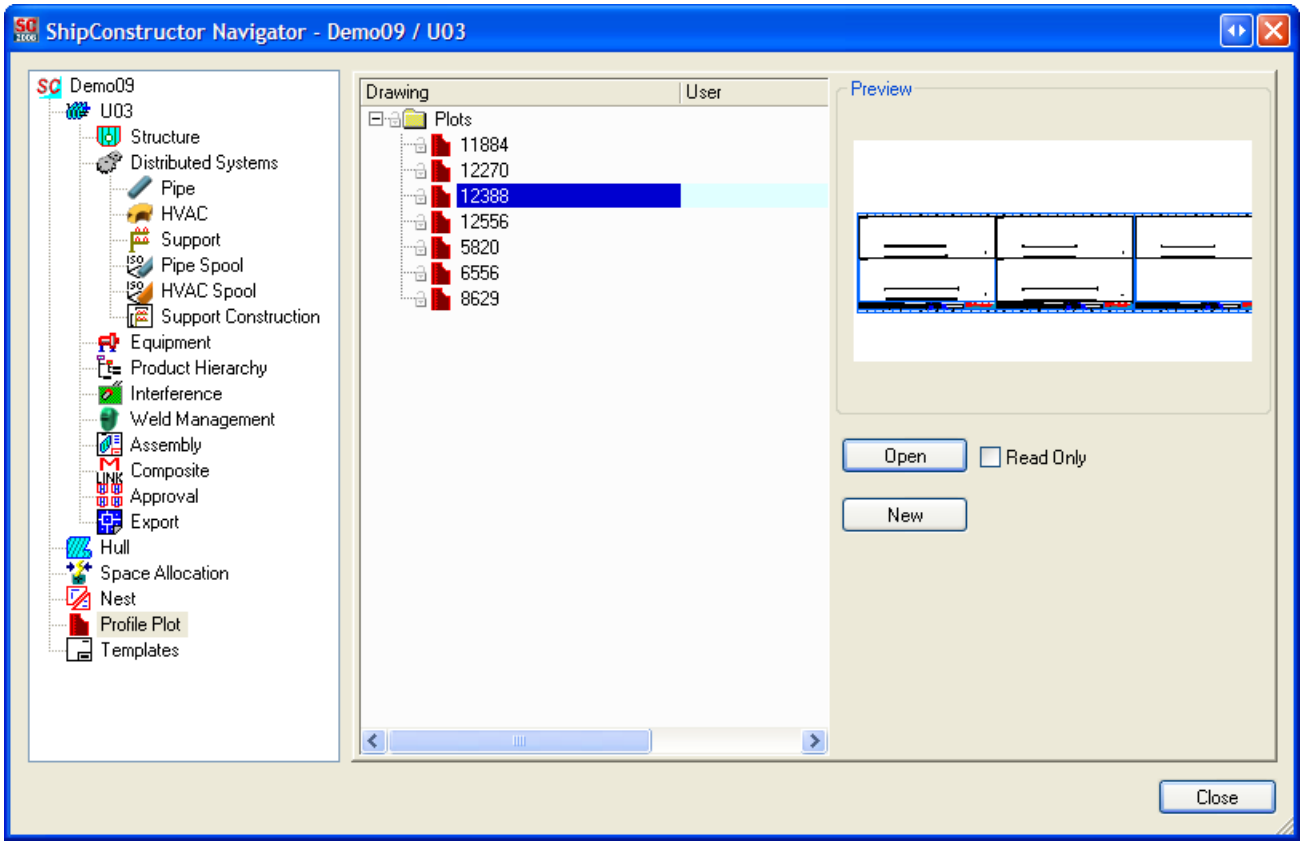
   Any legacy keywords in the template drawing have been replaced with the corresponding ShipConstructor keywords.

Create a Profile Plot Drawing

When you create a profile plot drawing, it is based on the profile plot drawing template (if there is one). If there is no profile plot drawing template, then creating a profile plot drawing creates a blank drawing.

To create a profile plot drawing

1. Choose ShipConstructor > Navigator (page 368) to open Navigator.
2. Select the Profile Plot page.

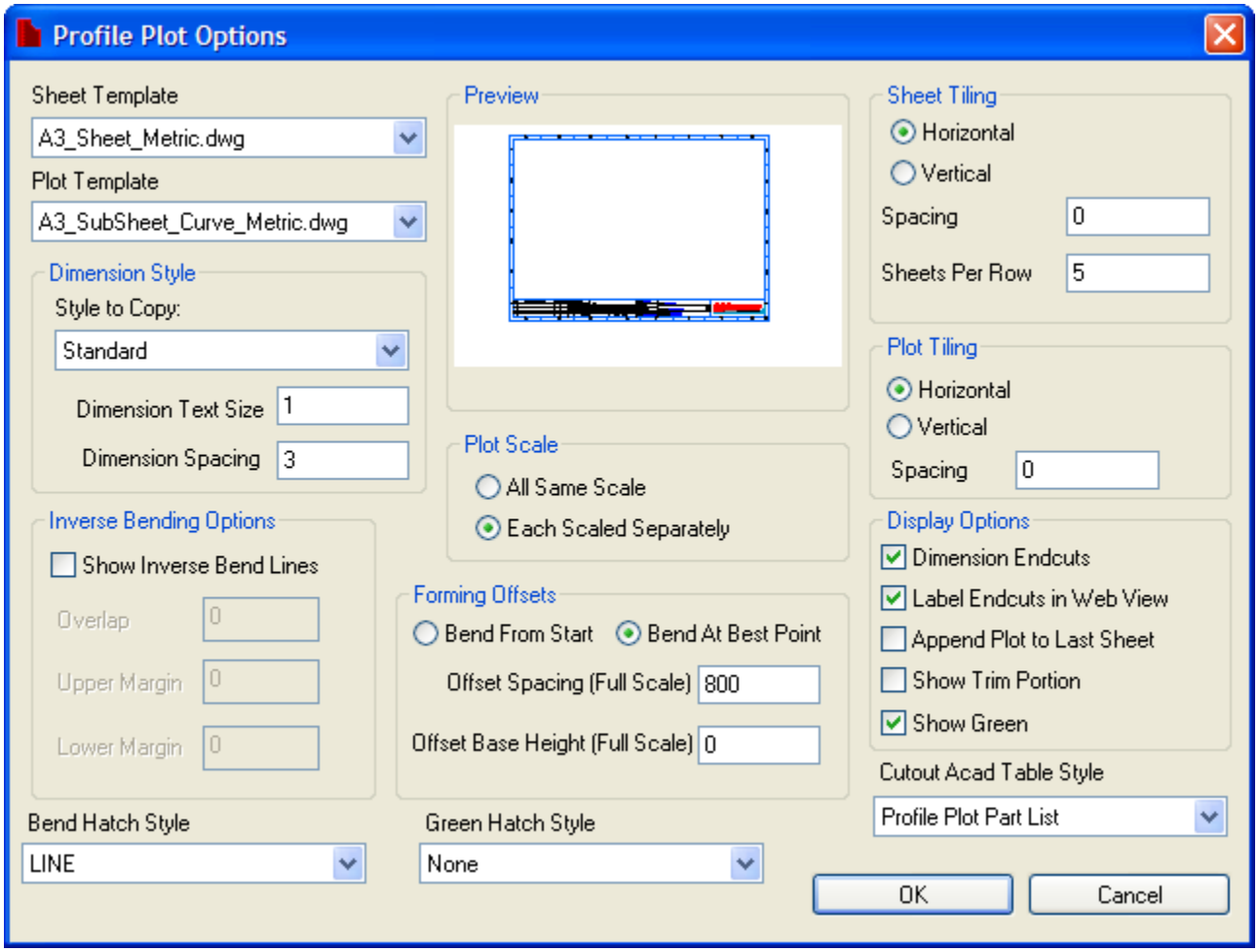


3. Click New.
4. Enter a File name.
5. Click OK.

Set Up Profile Plots

To set up profile plots

1. Choose SC Profile Plots > Options (page 532) to open the Profile Plot Options window.

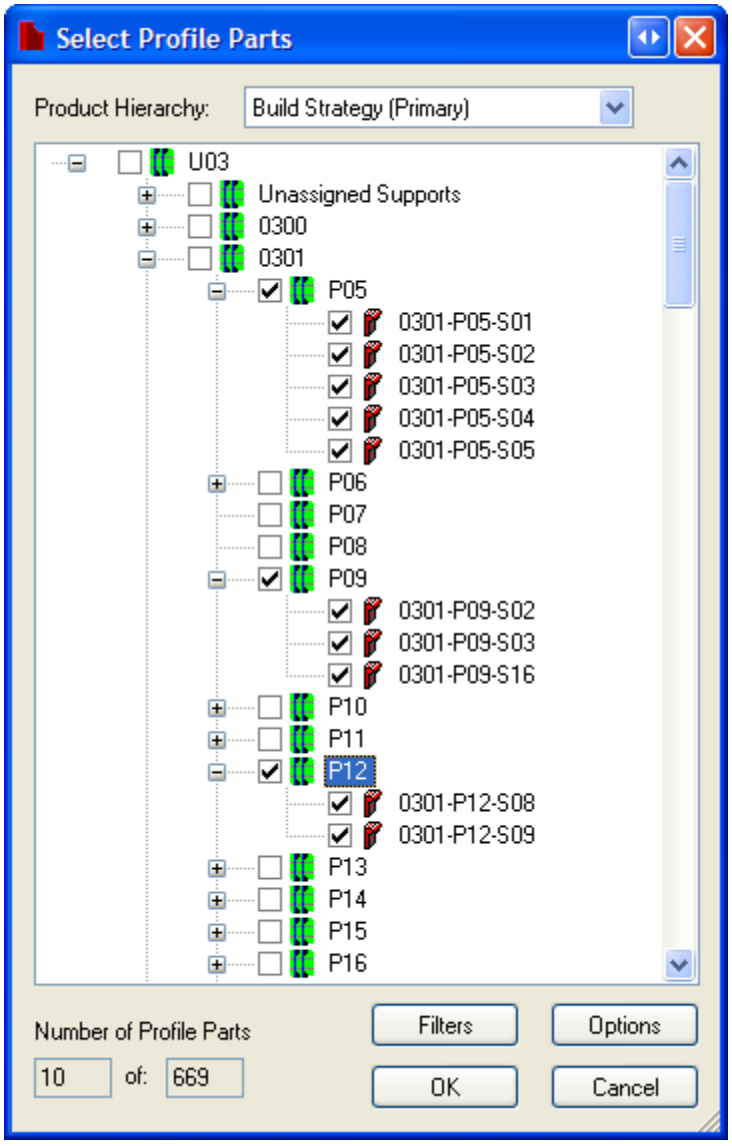


2. Set the desired properties. See Profile Plot Options Window (page 533).
3. Click OK.

Insert Profile Plots Into a Profile Plot Drawing

To insert a profile plot into a profile plot drawing

1. Choose SC Profile Plots > Insert Profile Plots (page 529) to open the Select Stiffeners window.

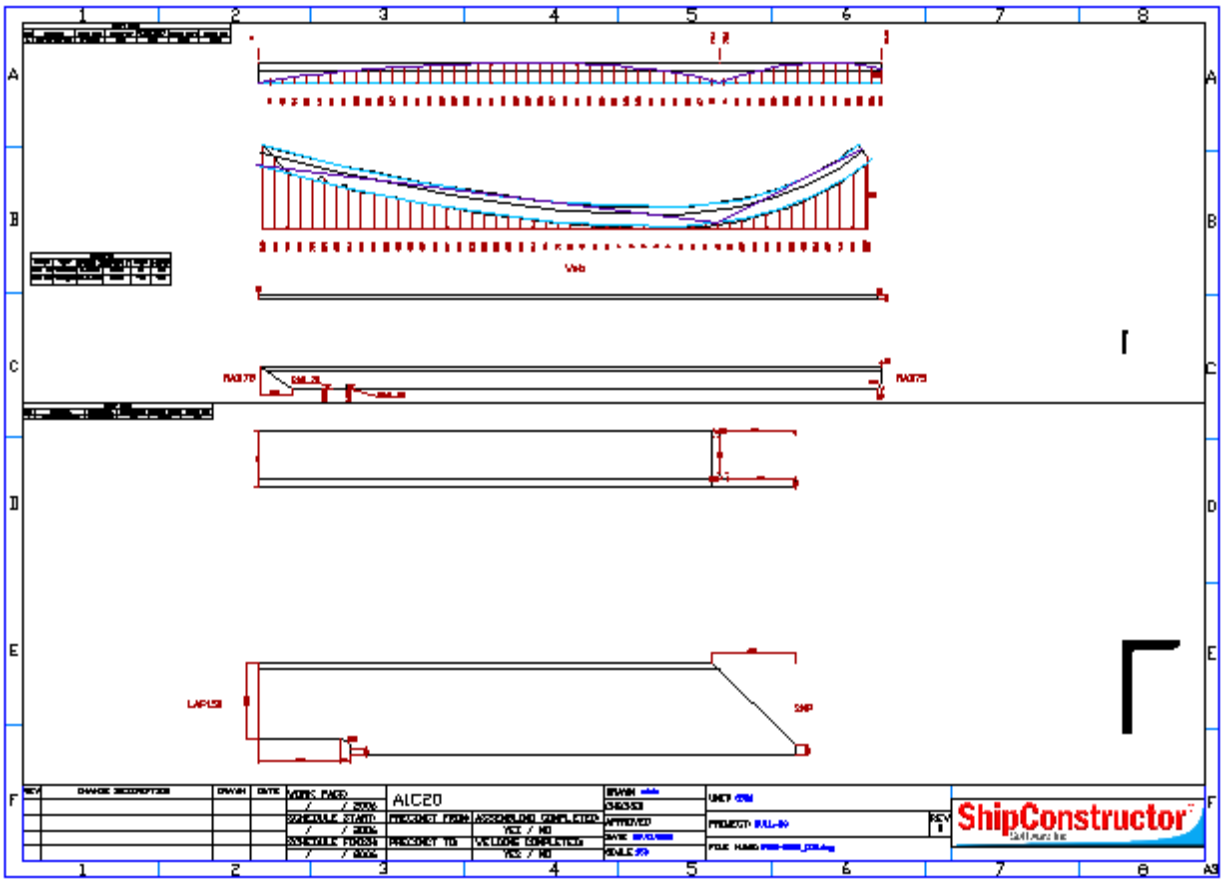


2. If you change the value in the Product Hierarchy dropdownlist, the assembly tree updates to show the assemblies for that product hierarchy.
3. Click the Filters button to open the Stiffener Filtering Window (page 531). Set any desired part filters and click OK
4. Click the Options button to open the Profile Plot Options Window (page 533). Set the desired plot options and click OK.
5. Click to place a check box beside the items you want to create profile plots for.
6. Click OK to create the profile plots.

When multiple profile parts are inserted into a profile plot drawing during the same insertion operation they may be grouped into the same plot based on their stock, endcuts, bend information, trims and cutouts. Identical parts are always grouped in the same plot. Parts will not be grouped together in following cases:

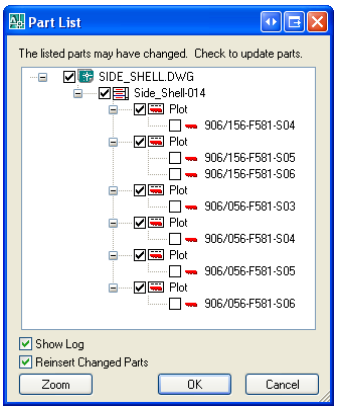
1. Cutouts create geometry differences on the stiffeners (cutouts have different shapes, oriented differently).
2. Matching cutouts are different distances from the start of the stiffener.
3. Trims are at different angles.
4. There are intersections between trims and cutouts.

In current version, green is not considered in parts grouping. This means parts will be grouped if one has green and one does not. It is random to which one it will show in the plot.



Updating Profile Plots

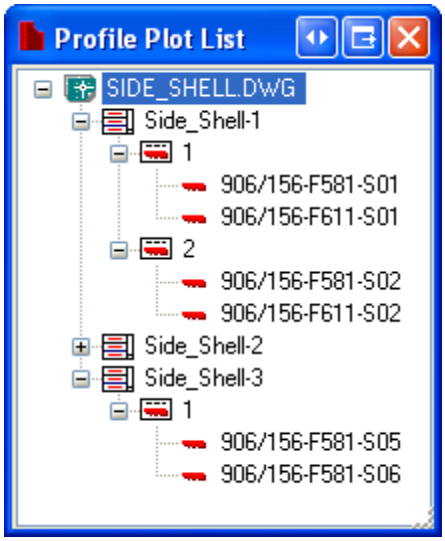
When you open a profile plot drawing that contains profile parts that have been changed in any way the Profile Plot Update Window will appear informing the user which profile plots need to be updated. If you choose to update a profile plot the plot and the plot's BOM will be updated automatically and any major changes made to the plot be entered in a log file. If you choose not to update a profile part you will be prompted again the next time you open the drawing. See Profile Plot Update Window (page 532).



Zoom Into a Profile Plot within a Profile Plot Drawing

To zoom in to a profile plot within a profile plot drawing

1. Choose SC Profile Plots > Zoom (page 536) to open the Profile Plot List window.



2. Double click on the drawing, sheet, plot or the part to zoom the view to.

Print Profile Plots

You can print all or individually selected plots on a printer.

To print profile plots

1. Choose SC Profile Plots > Print Plots (page 537) to open the Print Plots window.
2. Set the desired properties. See Print Plots Window (page 538).
3. Select the profile plots you want to print.
4. Click OK.

Check Profile Plots

You should periodically check your profile plots to make sure the project database and the profile plot drawings are synchronized. For example, if someone manually deletes a profile plot drawing (without using ShipConstructor), then the project database and the profile plot drawings no longer correlate to each other exactly. When you check your profile plots, ShipConstructor automatically creates any missing profile plots.

To check profile plots

1. Choose SC Profile Plots > Check Profile Drawings (page 538) to open the Update Options window.
2. Set the desired options. See Update Options Window (page 539).
3. Click OK.

Update Profile Plot BOMs

When a profile plot contains a stiffener that has been edited or changed in some way the BOM of the affected profile plots will need to be updated so that they reflect the updated information from the plotted parts.

To update all the BOMs in a profile plot drawing

1. Choose SC Profile Plots > Update All BOMs (page 539)

   All BOMs within the current profile plot drawing will be updated with the latest information from the plotted parts.

Profile Nesting

Profile nesting is the process of efficiently arranging profile parts onto raw profile stock.

There are three states that a nest can have:

- Created – When you create a nest, you can add and remove parts to and from the nest. You can also make changes to parts within the nest.
- Issued – When a nest is finalized and you do not intend to make changes to the parts within it, you issue the nest. You cannot modify the parts within an issued nest. Issuing a nest means you make it available for production to cut.
- Cut – After production has cut the nest on profile stock, you change the nest's status to cut. This indicates that the parts have been cut and cannot be modified.

Set Up Profile Stocks

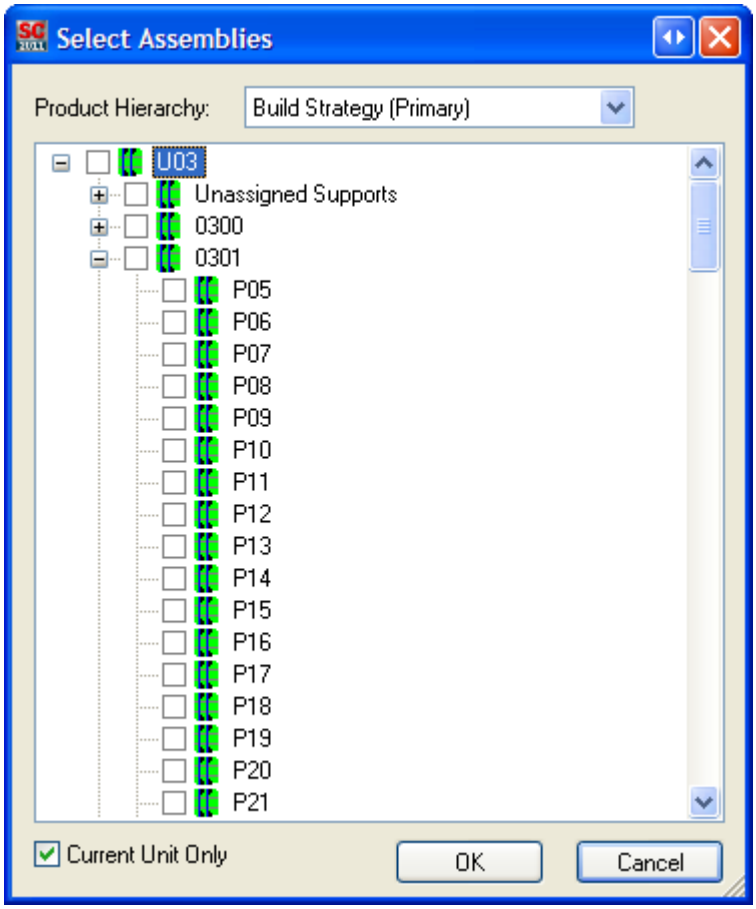
You must enter the lengths and quantities of each stock to be able to nest stiffeners and faceplates on stock lengths.

You will also need to set up a naming convention for the profile nests. See Naming Conventions (page 35).

Select an Assembly and Stock for Profile Nesting

To select an assembly and stock for profile nesting

1. Choose ShipConstructor > Profile Nesting (page 386) to open the Profile Nest – Select Asssemblies window.

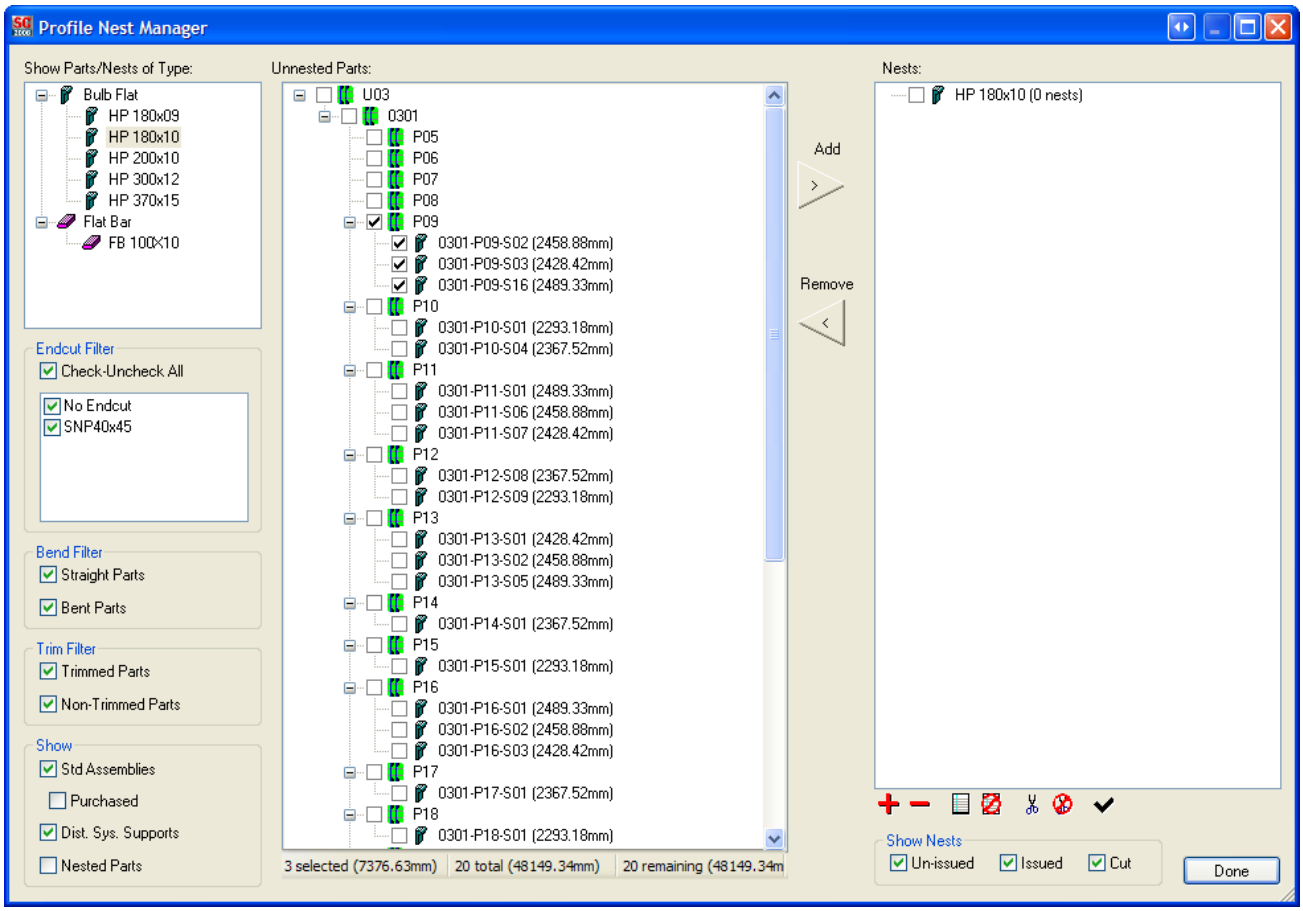


2. Click the check box for the assemblies you want to include in profile nesting. If you change the value in the Product Hierarchy dropdownlist, the assembly tree updates to show the assemblies for that product hierarchy.

   Current Unit Only – This option minimizes the amount of information in the tree, speeding up the window, and is especially useful for large projects. This only applies to the primary product hierarchy. For non-primary product hierarchies the entire project is shown.

3. Click OK.

   The Profile Nest Manager window appears.



4. In the Show Parts/Nests of Type list, select the stock you want to nest.

   The Unnested Parts window lists all unnested parts.

   The Nests window lists all nests for the selected stock. Initially, there will be no nests listed here.

   Note: In this example, the flatbar stock is selected. Flatbar parts that have been nested on plate stock are not listed.

Create or Delete Profile Nests or Remnants

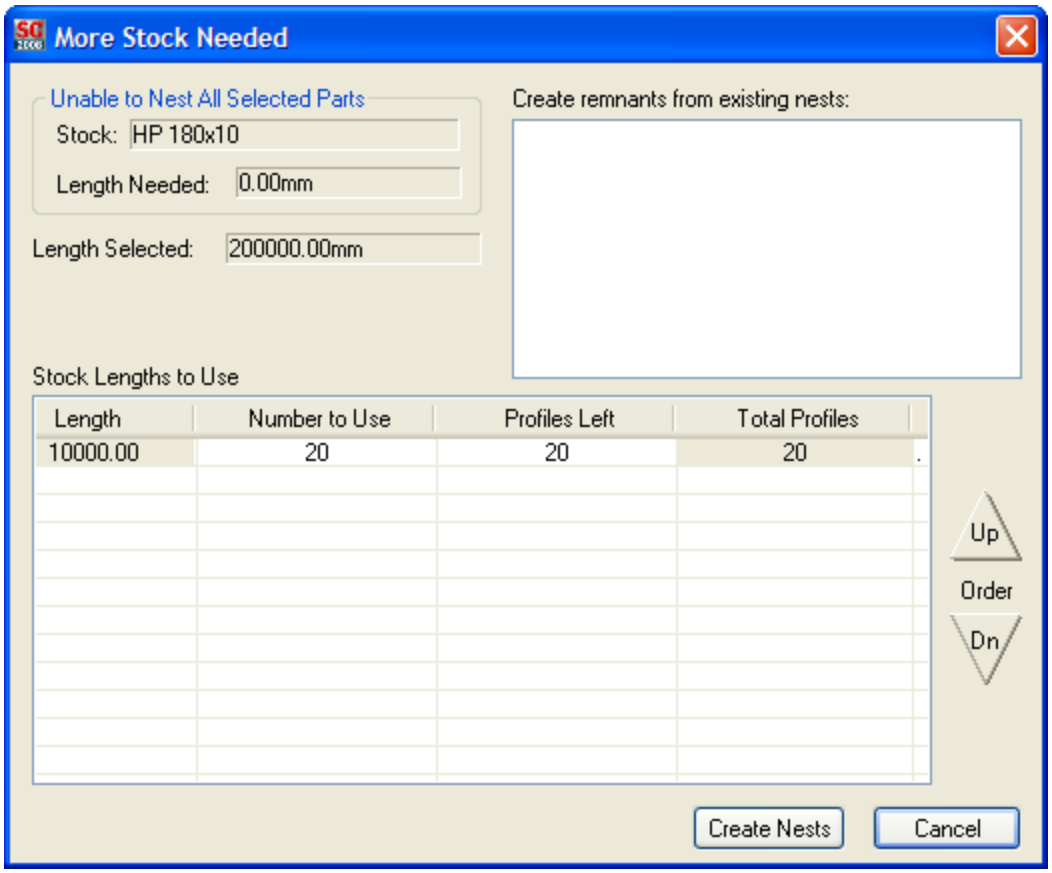
There are two ways you can create profile nests or remnants:

- Create empty profile nests or remnants (see Create Empty Profile Nests or Remnants (page 307)) and then later add parts to them (see Add and Remove Parts to and from Profile Nests (page 309)).
- Create profile nests or remnants for selected parts (see Automatically Create Profile Nests for Selected Parts (page 310)).

Create Empty Profile Nests or Remnants

To create empty profile nests or remnants

1. See Select an Assembly and Stock for Profile Nesting (page 305) to reach the Profile Nest Manager. In this window, click the (plus) button to open the Profile Nest window.



Each stock length is listed in the table (Length) with the available quantity (Profiles Left) and total quantity (Total Profiles).

Note: If there are any issued nests, they are listed under Create remnants from existing nests. To create a remnant, select the check box beside the issued nest.

2. For each stock length, enter the number of items to use (Number to Use) for the new nest.

   Length Selected is the total length of nests to create.

3. Click Create Nests.

   The new nests appear under Nests in the Profile Nest Manager.

   ShipConstructor automatically generates profile nest names based on the stock name. For example, if the stock name is FB100x12, the profile nest names will be FB100x12-0001, FB100x12-0002, and so on.

   Remnant nest names are based on the name of the nest that the remnant originated from. This lets you track the stock that a remnant is created from. For example, remnants created from the nest name FB100x12-0001 will be named FB100x12-0001-R1, FB100x12-0001-R2, and so on.

Delete a Profile Nest or Remnant

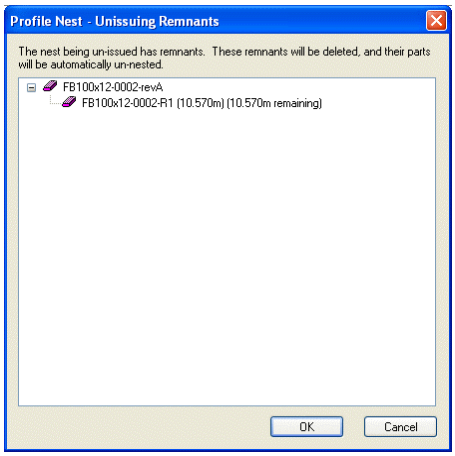
You can delete a profile nest or remnant that you no longer need as long as the following conditions are met:

- The nest must be un-cut.
- The nest must be un-issued.
- The nest must have no remnants.
- The nest must be free of nest parts.

To delete a profile nest

1. See Select an Assembly and Stock for Profile Nesting (page 305) to reach the Profile Nest Manager. In this window, under Nests, select the nests you want to delete.
2. To delete the nests, do either of the following:
   - Click (minus).
   - Right-click one of the selected nests and choose Delete Nest.

   If the nest has remnants, the Profile Nest – Unissuing Remnants window appears, listing the nest, remnants, and parts, and letting you delete the nest and its remnants at once.


Profile Nest - Unissuing Remnants
The nest being un-issued has remnants. These remnants will be deleted, and their parts will be automatically un-nested.
FB100x12-0002-revA
FB100x12-0002-R1 (10.570m) (10.570m remaining)
OK
Cancel

Add and Remove Parts to and from Profile Nests

You can either manually add parts to existing profile nests, or you can automatically create profile nests for selected parts.

Manually Add Parts to Profile Nests

You can add individual parts to individual profile nests, or you can add multiple parts to multiple nests. If you are adding multiple parts to multiple nests, ShipConstructor determines the optimal nesting configuration for the parts. ShipConstructor can nest thousands of parts in a few seconds. Nesting performance improves when large numbers of parts are nested simultaneously.

To add parts to profile nests

1. In the Profile Nest Manager, under Unnested Parts, click the check box for each part to add to nests.

   Note: You can filter the list of Unnested Parts by endcut type (Endcut Filter), bend type (Bend Filter), or trimmed and non-trimmed parts (Trim Filter).

   Checking the check box for an assembly will select all of the parts within the assembly.

   The bottom of the Unnested Parts list displays three things:

   - The selected parts and their combined length
   - The total number of parts of the selected stock and their combined length
   - The number of un-nested parts of the selected stock and their combined length
2. In the Nests list, click the check box for the nests to add parts to.

   Note: You can filter the list of Nests by UnIssued, Issued, or Cut.

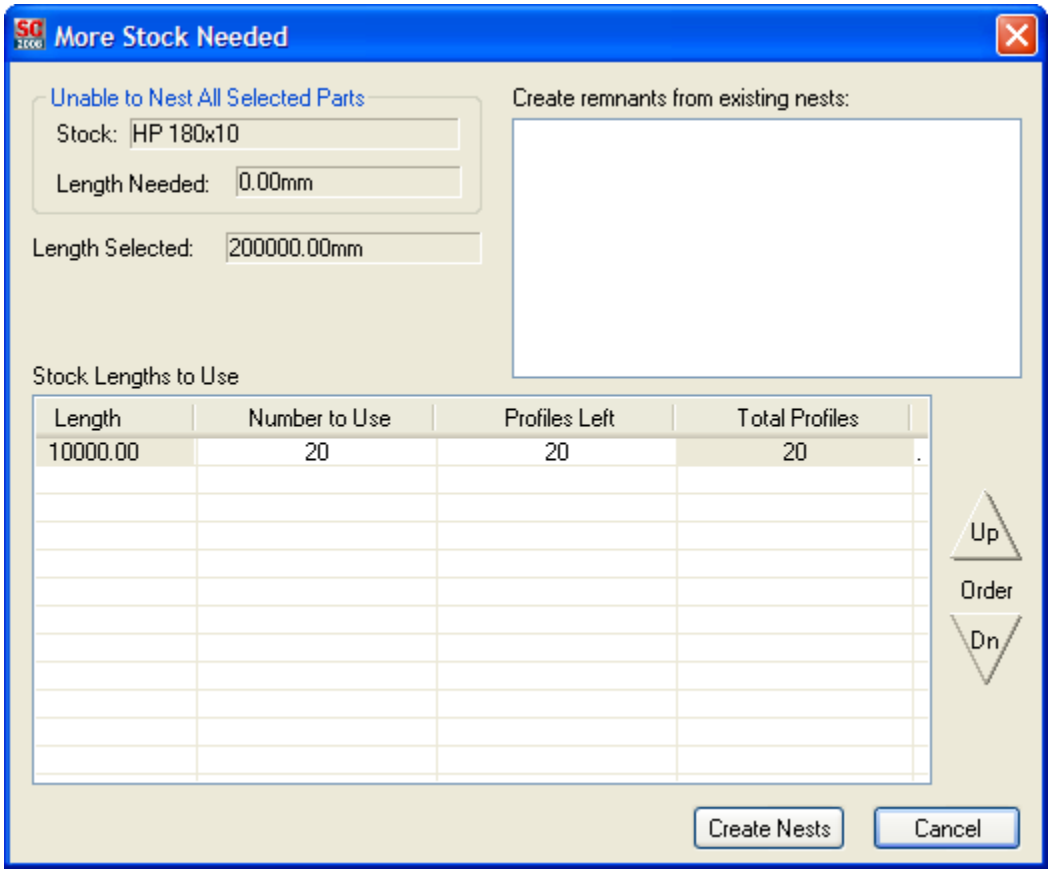
3. Click (Add) to add the selected parts to the selected nests.

Note: If there is not enough length within the selected nests for the selected parts, ShipConstructor nests as many parts as possible and then opens the Profile Nest – More Stock Needed window for the remaining parts.

Automatically Create Profile Nests for Selected Parts

To automatically create profile nests for selected parts

1. In the Profile Nest Manager under Unnested Parts, click the check box for each part you want to nest.
2. Make sure there are no nests selected under Nests.
3. Click (Add) to open the Profile Nest window.



Length Needed is the length of the selected parts, including the linear part gap.

Each stock length and remnant is listed in the table (Length) with the available quantity (Profiles Left) and total quantity (Total Profiles).

Note: If there are any issued nests, they are listed under Create remnants from existing nests. To create a remnant, select the check box beside the issued nest.

4. For each stock length or remnant, enter the number of items to use (Number to Use) for the new nest.

   Length Selected is the total length of nests you want to create.

   Note: Even if Length Selected is larger than Length Needed, the selected parts still may not fit onto the selected stock.

5. Click Create Nests to create the nests and add the selected parts to them.

Note: If more nests are selected than are needed to nest the selected parts, ShipConstructor will not create the empty nests. If you have not allocated enough stock for the selected parts, the Profile Nest window re-appears, letting you increase the amount of stock to use.

The new nests appear under Nests in the Profile Nest Manager.

Remove Parts From Profile Nests

You can remove parts from profile nests that have not been issued or cut. When you remove parts from profile nests, the parts are returned to the list of parts available for nesting and will appear in the un-nested parts list.

To remove parts from profile nests

1. In the Profile Nest Manager, under Nests, click the check box for each part you want to remove.

   Checking the check box for a nest will select all parts within the nest.

   Note: You can filter the list of Nests by UnIssued, Issued, or Cut.

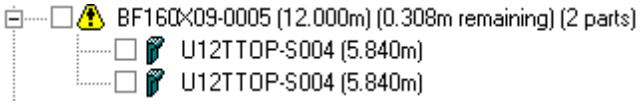
2. Click (Remove) to remove the selected parts from nests.

   ShipConstructor updates the lengths for the nests and creates a record for un-nesting each part.

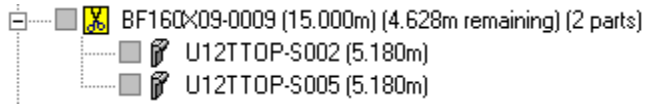
Changing Parts and Stocks and Checking Profile Nests

When anyone edits a part, ShipConstructor performs a check to see if the part has been nested. If so, changes to the part may affect the nest.

For example, after you nest a part, it is possible that someone else may change the part before you issue the nest. Potentially problematic changes include changing part length, endcuts, or stock type. If any part properties are changed after the part is nested, the part's nest is listed with a warning icon. For uncut nests, the warning icon is an exclamation mark within a yellow triangle.



For cut nests, the warning icon is a pair of scissors within a yellow square.



If an error message appears, run a nest check on the nest.

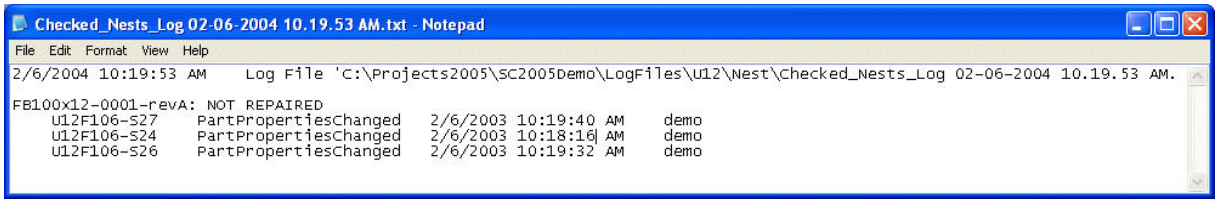
Similarly, if someone changes the profile stock settings for the Smallest Remnant or the Kerf, you must run a nest check for the nests of this stock. For example, if the nesting gap is increased, the amount of space required on the nest increases, and parts may no longer fit onto the nest.

If someone tries to delete a nested part, ShipConstructor checks if the nest has been issued or cut. If the nest has been issued or cut, the part cannot be deleted. To delete the part, first un-cut and un-issue the nest, then un-nest the part, and finally delete the part in the drawing.

To check nests

1. In the Profile Nest Manager, under Nests, do either of the following:
   - To check one nest, right-click on the nest, and choose Check Nest.
   - To check all nests listed, click (Check All Visible Nests).

   ShipConstructor checks if any parts have been changed, ensures that the parts still fit on the nests and their remnants, and checks if the stock of any of the parts has been changed. If any part properties have changed, ShipConstructor displays a text file containing a list of the changes.


Checked_Nests_Log 02-06-2004 10.19.53 AM.txt - Notepad
File Edit Format View Help
2/6/2004 10:19:53 AM Log File 'C:\Projects2005\SC2005Demo\LogFiles\U12\Nest\Checked_Nests_Log 02-06-2004 10.19.53 AM.
FB100x12-0001-revA: NOT REPAIRED
U12F106-S27 PartPropertiesChanged 2/6/2003 10:19:40 AM demo
U12F106-S24 PartPropertiesChanged 2/6/2003 10:18:16 AM demo
U12F106-S26 PartPropertiesChanged 2/6/2003 10:19:32 AM demo

If the parts have been fixed, the nest icon will change and appear as fixed.

Issue and Un-issue Profile Nests

Once you are satisfied with the arrangement of parts in a profile nest, you can issue the nest. Issuing a nest means you make it available for production to cut. You cannot change any parts in an issued nest (unless you un-issue the nest).

You can issue any number of nests simultaneously. To issue a nest, the nest must contain at least one part.

You can also un-issue a nest.

Issue Profile Nests

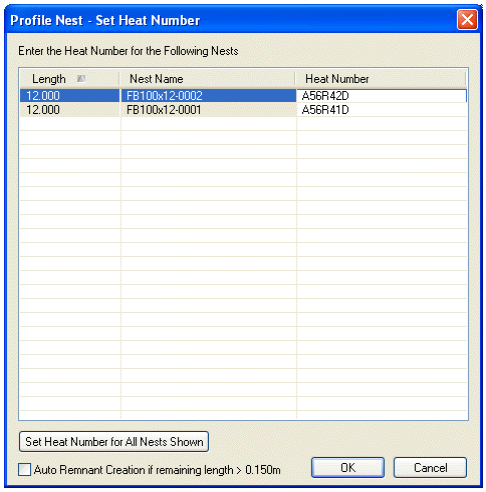
To issue profile nests

1. In the Profile Nest Manager, under Nests, click the check box for each nest you want to issue.

   Note: You can filter the list of Nests by UnIssued, Issued, or Cut.

2. Click (Issue Selected Nests).

   The Profile Heat – Set Heat Number window appears, listing the nests that you chose to issue.



3. You can set a Heat Number for each nest for tracking and quality assurance purposes. To give all nests being issued the same heat number, click Set Heat Number for All Nests Shown.
4. To automatically create a remnant from the remaining material (if it is longer than the Smallest Remnant for that stock), select Auto Remnant Creation if remaining length > #. If you do not select this option, the remaining material becomes scrap that you can manually create a remnant from later.
5. Click OK to issue the nests.

   Issued nests appear with a gray check box in the Profile Nest Manager.

You cannot add or remove parts to or from issued nests.

Once you have issued a nest, you can create remnants from it (see Create or Delete Profile Nests or Remnants (page 307)).

To issue profile nests and create remnants

1. In the Profile Nest Manager, under Nests, select the nest you want to issue.
2. Right-click one of the selected nests and choose Create Remnant.

To change the heat number of a profile nest

1. In the Profile Nest Manager, under Nests, select the nests you want to change. To select multiple nests, hold the Shift or Ctrl key while clicking.
2. Right-click one of the selected nests and choose Change Heat Number.

Un-issue Profile Nests

You may need to un-issue a profile nest if you want to make changes to the parts within it. (You cannot modify parts within an issued nest.)

To un-issue profile nests

1. In the Profile Nest Manager, under Nests, click the check box for each nest you want to un-issue.

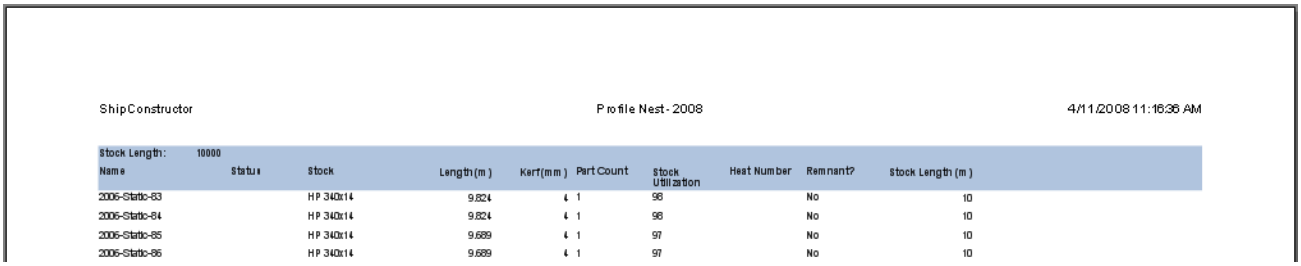
   Note: You can filter the list of Nests by UnIssued, Issued, or Cut.

2. Click (Un-issue Selected Nests).

Generate Profile Nest Reports

You can create profile nest reports to provide for production. There are three kinds of profile nest reports:

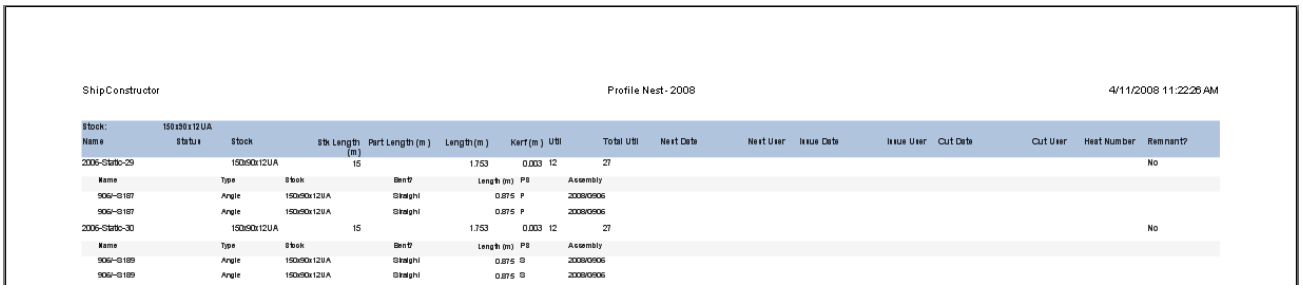
Summary report – Includes all nest information but does not include a listing of parts within the nests.


ShipConstructor Profile Nest - 2008 4/11/2008 11:16:36 AM

Stock Length: 10000

Name	Status	Stock	Length (m)	Kerf (mm)	Part Count	Stock Utilization	Heat Number	Remnant?	Stock Length (m)
2006-Static-83		HP 340x14	9.824	4	1	98		No	10
2006-Static-84		HP 340x14	9.824	4	1	98		No	10
2006-Static-85		HP 340x14	9.689	4	1	97		No	10
2006-Static-86		HP 340x14	9.689	4	1	97		No	10

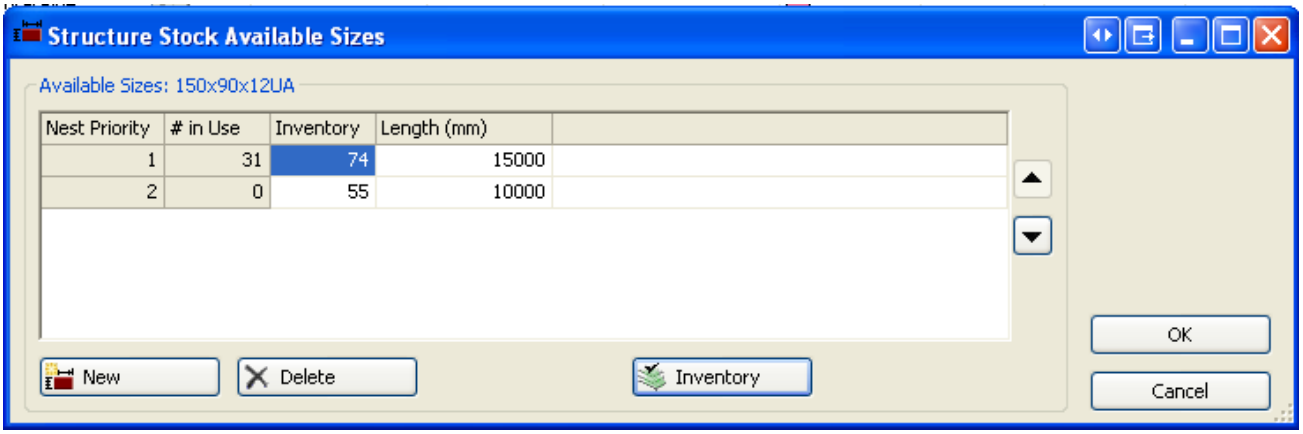
Detailed report – Includes all nest information and all part information.


ShipConstructor Profile Nest - 2008 4/11/2008 11:22:26 AM

Stock: 150x90x12UA

Name	Status	Stock	Stk Length (m)	Part Length (m)	Length (m)	Kerf (m)	Util	Total Util	Nest Date	Nest User	Issue Date	Issue User	Cut Date	Cut User	Heat Number	Remnant?
2006-Static-29		150x90x12UA	15		1.753	0.003	12	27								No
Name	Type	Stock	Bent?		Length (m)	PS	Assembly									
906/-S187	Angle	150x90x12UA	Straight		0.875	P	2008/0906									
906/-S187	Angle	150x90x12UA	Straight		0.875	P	2008/0906									
2006-Static-30		150x90x12UA	15		1.753	0.003	12	27								No
Name	Type	Stock	Bent?		Length (m)	PS	Assembly									
906/-S189	Angle	150x90x12UA	Straight		0.875	S	2008/0906									
906/-S189	Angle	150x90x12UA	Straight		0.875	S	2008/0906									

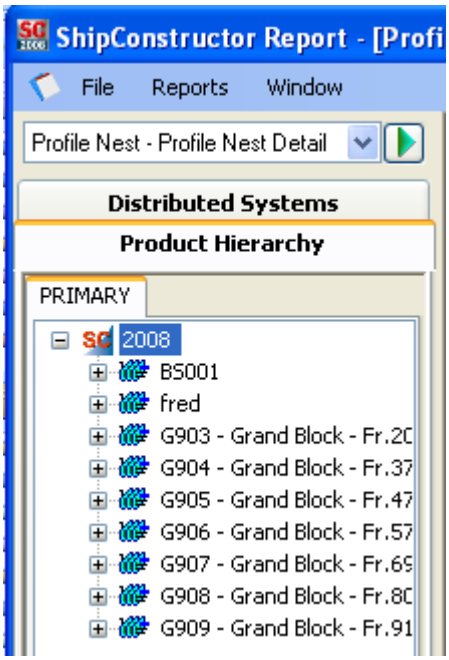
Profile stock usage report – Shows how many lengths of a certain stock have been assigned to the project (Inventory) and how many nests have been created for a particular length of stock (# in Use).



Generate a Summary Report or Detailed Report

To generate a summary report or detailed report

In Report, choose either Profile Nest – Profile Nest Detail or Profile Nest – Profile Nest Summary from the report type droplist, then click Generate Report to generate a report for the selected Product Hierarchy. If this report type is not available, you can import it from the Imperial/Metric Project templates provided with ShipConstructor.

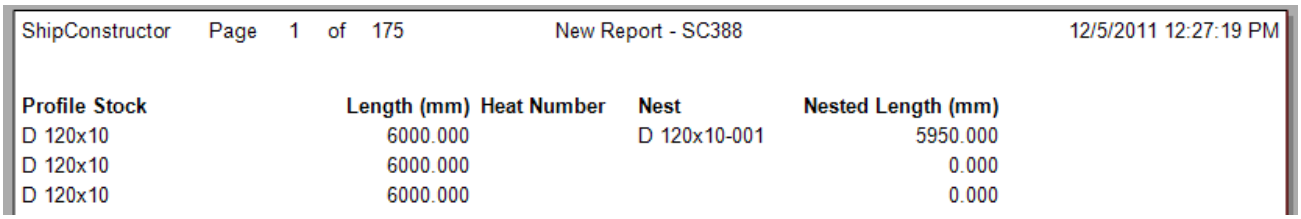


If you would like to modify the layout or columns shown in the Profile Nest reports, this can be done using the Report Definition Editor. Please see the Reports documentation for more details on editing report definitions.

Generate a Stock Usage Report

To generate a stock usage report

In Report, create a new report definition under Other – Stock Inventory. To this report definition, add the Profile Stock Inventory report section, and include the desired fields. Choose your new report definition from the drop list, then click Generate Report to generate a report for all of your profile inventory items.

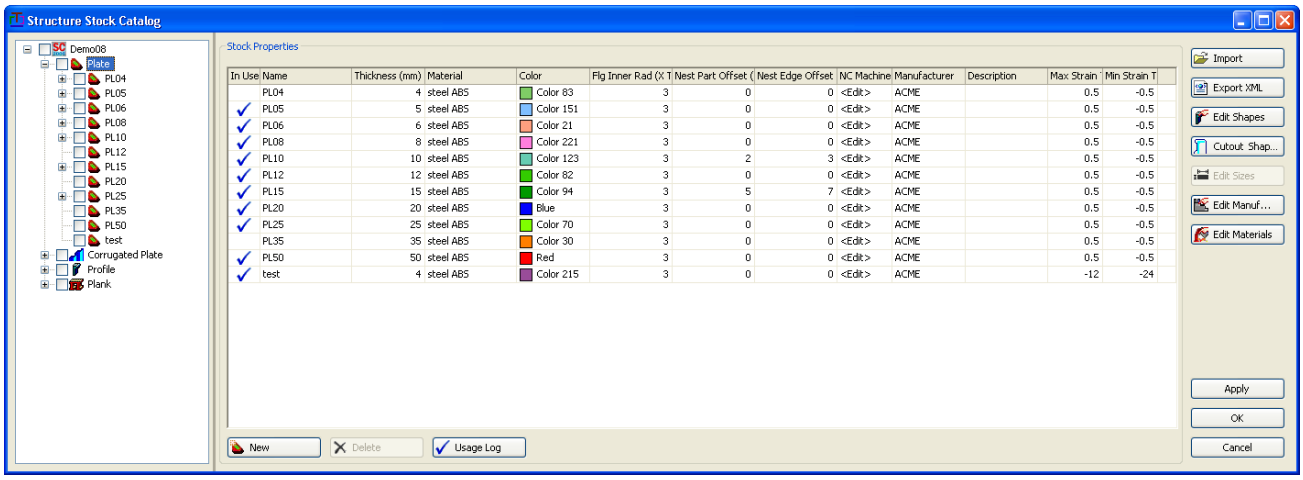


ShipConstructor Page 1 of 175 New Report - SC388 12/5/2011 12:27:19 PM

Profile Stock	Length (mm)	Heat Number	Nest	Nested Length (mm)
D 120x10	6000.000		D 120x10-001	5950.000
D 120x10	6000.000			0.000
D 120x10	6000.000			0.000

To view stock usage for a particular stock

1. In Manager, choose Structure > Stock Catalog to open the Stock Catalog Editor.


Structure Stock Catalog
Demo08
Plate
PL04
PL05
PL06
PL08
PL10
PL12
PL15
PL20
PL25
PL35
PL50
test
Corrugated Plate
Profile
Plank
Stock Properties
In Use Name Thickness (mm) Material Color Fig Inner Rad (X T Nest Part Offset (Nest Edge Offset NC Machine Manufacturer Description Max Strain Min Strain T
PL04 4 steel ABS Color 83 3 0 0 <Edit> ACME 0.5 -0.5
PL05 5 steel ABS Color 151 3 0 0 <Edit> ACME 0.5 -0.5
PL06 6 steel ABS Color 21 3 0 0 <Edit> ACME 0.5 -0.5
PL08 8 steel ABS Color 221 3 0 0 <Edit> ACME 0.5 -0.5
PL10 10 steel ABS Color 123 3 2 3 <Edit> ACME 0.5 -0.5
PL12 12 steel ABS Color 82 3 0 0 <Edit> ACME 0.5 -0.5
PL15 15 steel ABS Color 94 3 5 7 <Edit> ACME 0.5 -0.5
PL20 20 steel ABS Blue 3 0 0 <Edit> ACME 0.5 -0.5
PL25 25 steel ABS Color 70 3 0 0 <Edit> ACME 0.5 -0.5
PL35 35 steel ABS Color 30 3 0 0 <Edit> ACME 0.5 -0.5
PL50 50 steel ABS Red 3 0 0 <Edit> ACME 0.5 -0.5
test 4 steel ABS Color 215 3 0 0 <Edit> ACME -12 -24
Import
Export XML
Edit Shapes
Cutout Shap...
Edit Sizes
Edit Manuf...
Edit Materials
New
Delete
Usage Log
Apply
OK
Cancel

2. Select the stock to include in the report.
3. Click Edit Sizes, and you will see the current project inventory and number in use.

Set Profile Nests to Cut or Un-cut

Once production has physically cut a nest, you should set the nest to cut in ShipConstructor.

Set Profile Nests to Cut

To set profile nests to cut

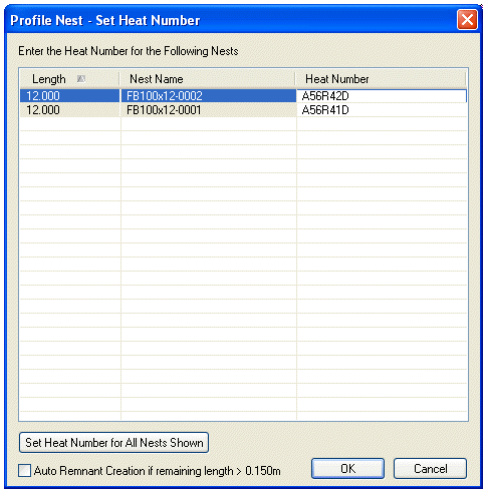
> Note: You cannot set a remnant to cut until the nest it came from has been set to cut.

1. In the Profile Nest Manager, under Nests, click the check box for each nest you want to set to cut.

   > Note: You can filter the list of Nests by UnIssued, Issued, or Cut.

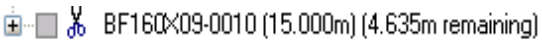
2. Click (Cut Selected Nests).

   The Profile Heat – Set Heat Number window appears, listing the nests you chose to issue.



See Issue Profile Nests (page 312) for details on the Profile Nest – Set Heat Number window. When you are finished with the list of nests, click OK.

The nests are now listed with a cut icon.



Set Profile Nests to Un-cut

To set profile nests to un-cut

1. In the Profile Nest Manager, under Nests, click the check box for each nest you want to set to un-cut.

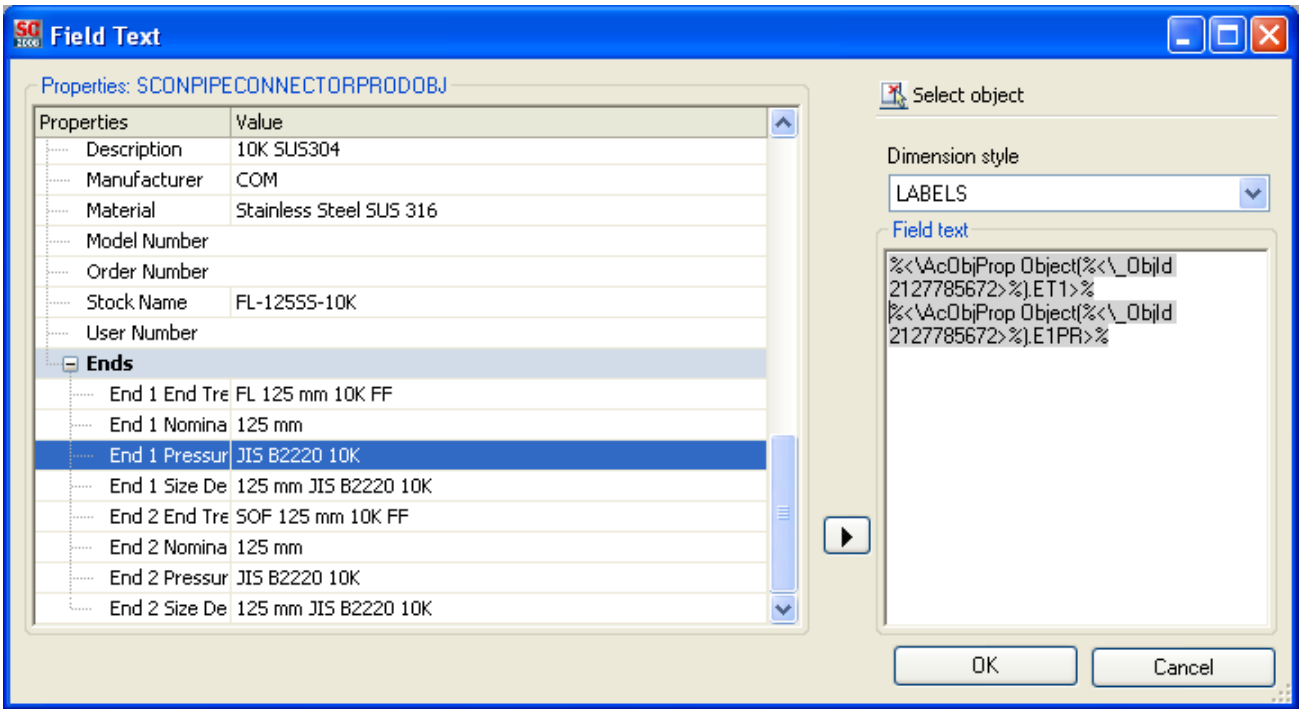
   Note: You can filter the list of Nests by UnIssued, Issued, or Cut.

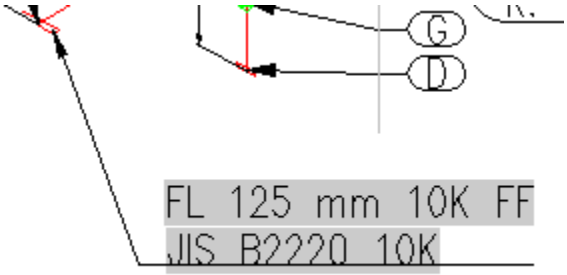
2. Click (Un-cut Selected Nests).

Part Property Labels

ShipConstructor supports extraction of part properties for display in production drawings. Information normally displayed in the AutoCAD properties window can be displayed in label form. ShipConstructor provides the means to select AutoCAD fields for insertion into a user editable MText block, that is then optionally attached to an AutoCAD label.

The MText block uses AutoCAD fields to extract information from ShipConstructor objects. For a thorough understanding of the AutoCAD fields and how to optionally specify formatting, see the AutoCAD help documents.





Access the Part Property Label function through SC Utilities > Property Label.

Properties
Lists the properties for the selected object. Values are shown for reference only and will be evaluated when the fields are updated.

Dimension Style
Selects the dimension style for the AutoCAD leader object.

Field Text
Double click the field or click the right arrow button to insert the field into Field text box. Insertion occurs at the current cursor position. You can add or edit the text inside the Field text box as needed.

The behavior of insertion is dependent on the cursor position with the field text. If the position is within a field, then the field will be replaced by the new one. If the cursor is not within a field then the field is inserted at the cursor position.

Select Object
Allows selection of other objects for field selection. This does not replace the object references currently inserted within the fields; rather it updates the field list. This allows for the creation of mtext blocks that reference multiple objects.

Placing Field Labels

To Place a Part Property Label

1. Choose SC Utilities > Property Labels > Property Label
2. Select the property that you wish to insert from the property list. Double click or use the right arrow button to insert the property into the field text box. The field is shown in raw field form, allowing the experienced user to edit the details directly.
3. Add any additional text as required to the label text and then click the OK button once complete
4. Select the leader origin.
5. Select the leader tail position. The MText will be attached once command is completed

> Note: Step 4 can be skipped if the ShipConstructor variable `SCSetSingleClickFieldLabel` is set to `<1>`. When the option is toggled on, then the first leader point is the item selection point. When labeling in Model Space the behavior may not be as desired the UCS projected selection point when labels are placed in Model Space.

Copying Field Labels

Field labels can be copied using standard AutoCAD techniques or by using the ShipConstructor's field copy commands.

Copying by AutoCAD techniques does not replace the object references within the fields. Another command must be run afterward if object references are to be changed. The ShipConstructor copy methods replace the object references during the copying procedure.

The SCFieldLabelCopy command copies the mtext block and then prompts the user to select a new object reference place and place a new leader section.

To Copy a Field Label / MText

1. Choose SC Utilities > Property Labels > Copy Property Label
2. Select the leader or mtext. ShipConstructor automatically determines the association between the label and mtext (if any)
3. Select the new part to replace the object reference within the field.
4. If the mtext is associated with a leader select the new leader origin.
5. Select the mtext or leader tail position.

The SCFieldLabelCopyQuick command copies the mtext block and then prompts the user to select a new object reference and position. The existing leader geometry is retained.

To Quick Copy a Field Label / MText

1. Choose SC Utilities > Property Labels > Quick Copy Property Label
2. Select the leader or mtext. ShipConstructor automatically determines the association between the label and mtext (if any).
3. Select the new part to replace the object reference within the field.
4. Select the leader or mtext position.

If AutoCAD methods are used to copy the label/mtext to a new position and you wish to replace the object reference within the field, use the SCReplaceObRefInField command to select a new object to use as reference. Note that all object references within the field will be replaced using this command and as a result, ill formed fields may result.

To Replace Object References within MText

1. Choose SC Utilities > Property Labels > Replace Object Reference
2. Select the mtext containing the fields.
3. Select the new object to 'point' the fields to.

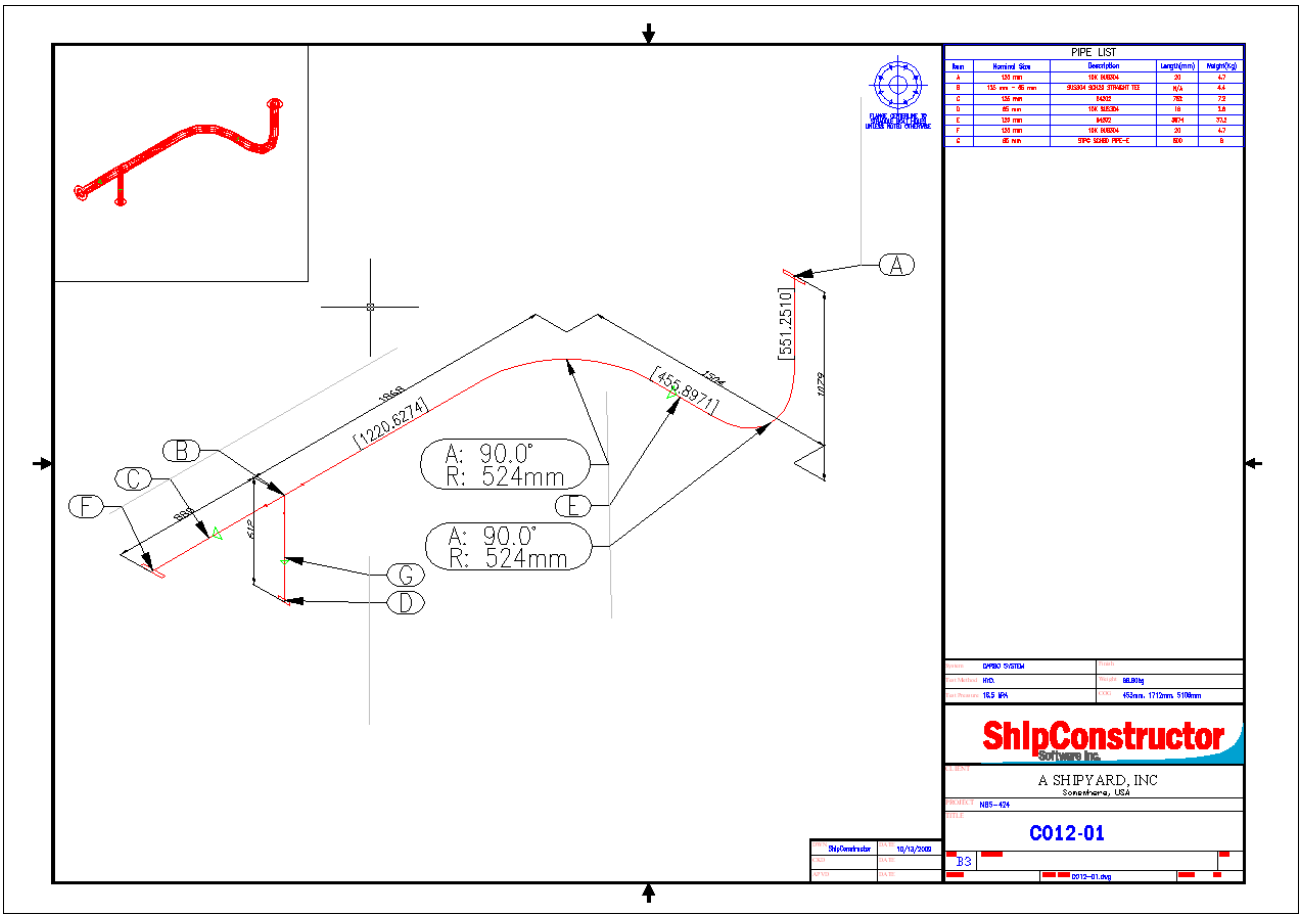
The AutoCAD effect of improperly evaluated fields is to display '####' in place of the field. You can display the field in its raw form and edit the field by using the SCEditField command.

To Edit Fields in MText

1. Choose SC Utilities > Property Labels > Edit Fields
2. Select the mtext containing the fields.
3. Edit the text within the Field text box.

Leader Distribution Lines

Leader distribution lines allow ShipConstructor labels to be automatically aligned and spaced. Leaders when attached to a distribution line, adjust as necessary when the line is moved, resized, or rotated. Leader distribution lines behave exactly like AutoCAD lines and therefore can be modified in the same manner as AutoCAD lines.



Distribution Lines – The leader distribution lines are visible as light gray lines in the above image. Distribution lines do not plot.

Leaders can be manipulated in several ways with respect to distribution lines:

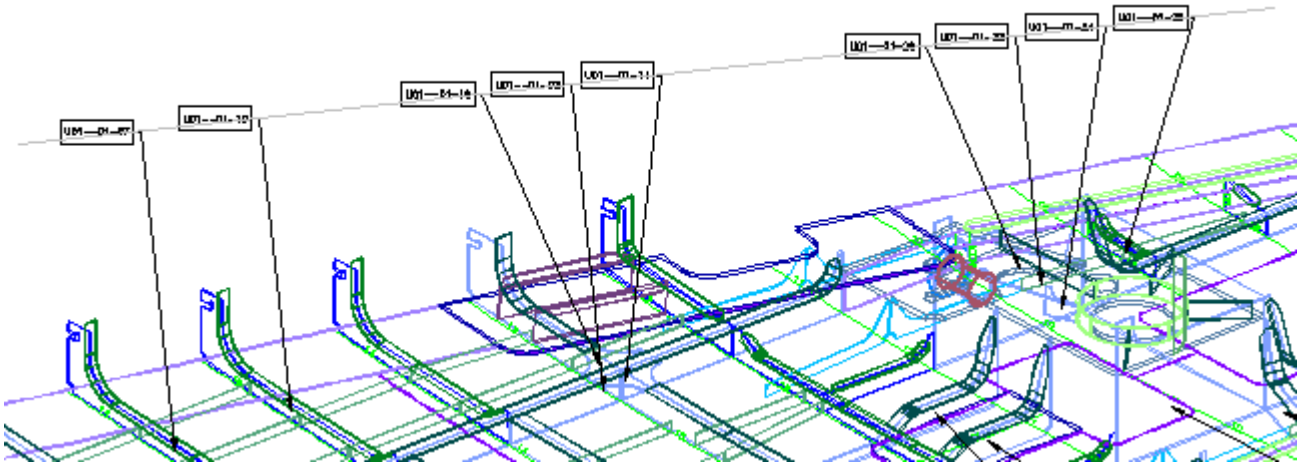
1. Dragged off a leader distribution line. The leader will be detached from the distribution line such that any further manipulation of the line will not affect the leader.
2. Repositioned along the distribution line. The leader's new position along the line is maintained even if the line is subsequently manipulated.
3. Dragged onto another distribution line. The leader is detached from the first line and attached to the new line.

4. Bulk transfer to another distribution line. Using the SCTRANSFERLEADERS command, the selected labels are transferred to another distribution line.
5. Distribution line manipulation. Transform the line using any of the standard AutoCAD methods. The labels will remain attached.

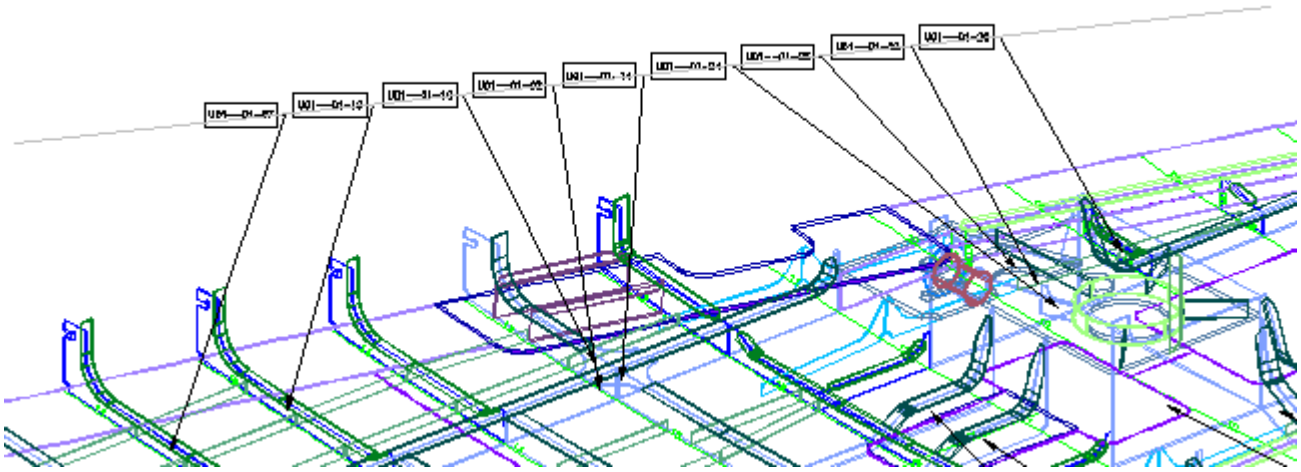
Note: In order for individual leader manipulations to be effective when positioning on distribution lines, enable the Nearest Osnap.

Line dependent variables:

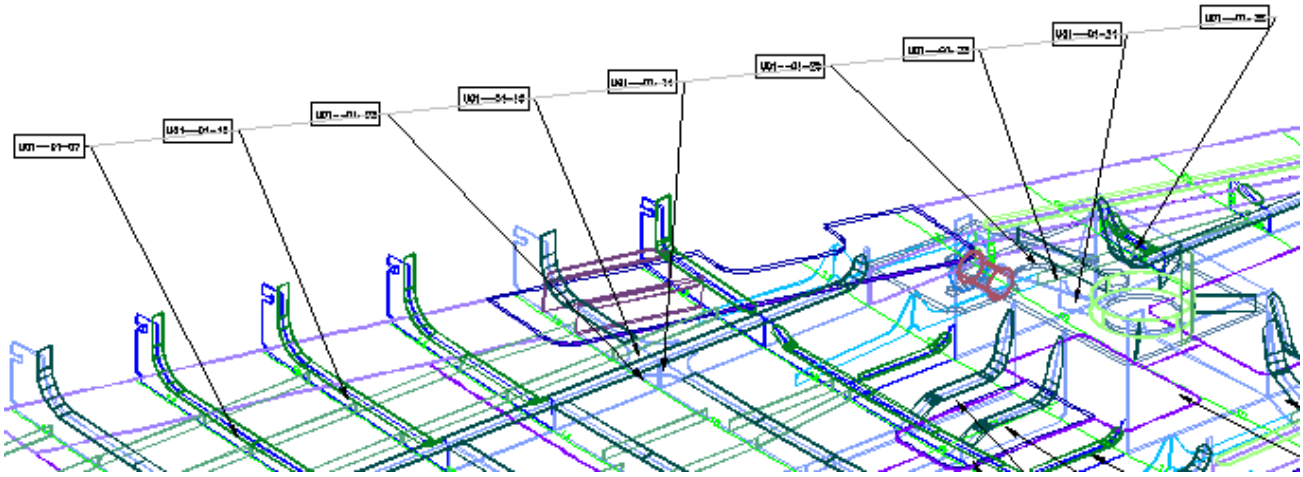
1. Bubble spacing - Bubble spacing can be set on a per-line basis. This forms the minimum spacing between the labels.
2. Leader attachment position – To ensure consistent label positioning, distribution lines enforce leader consistent insertion modes among all the leaders on the line.
3. Distribution mode – Can be:
    a. Nearest – Labels stay on positions along the line which are the cloest possible to their represented parts. Nearest mode is the default setting when a distribution line is created.



    b. Center – labels are centered in the middle of distribution lines.



    c. Equidistant – labels bubbles are evenly distributed on the line.



Set Leader Insertion Mode

This command is used to modify the insertion mode of all the labels connected to selected distribution lines.

To set the leader insertion mode

1. Choose SC Assembly > Set Leader Insertion Mode.

   ```
   Select distribution lines:
   ```

2. Select the distribution lines.

   ```
   Enter an insertion mode [Left/Right/Center/closest Horizontal] <H>:
   ```

3. Enter the new insertion mode.

Redistribute Leaders

This command is used to redistribute leaders on the distribution line in one of the three modes.

To Redistribute Leaders

4. Choose SC Assembly > Set Redistribute Leaders

   ```
   Select distribution lines:
   ```

5. Select the distribution lines.

   ```
   Enter a distribution mode [Center/Nearest/Equidistant] <Nearest>:
   ```

6. Enter one of the three distribution modes.

Export To Dwg

Export to Dwg generates a drawing that can be opened by AutoCAD without ShipConstructor being present on the computer.

The export process allows the user to configure the viewport display options that are used when generating the output entities. The default settings are read from the ShipConstructor source drawing's model space configuration. These options can be changed through the UI or the command line when performing an export.

There are a number of steps to export a ShipConstructor drawing to a regular .dwg that can be read by AutoCAD. First the original drawing is copied to a temporary location to avoid making changes to the source document. Then all of the xrefs

will be removed and a local copy of the data will be created in a block with the same name as the xref block. Once that is done all ShipConstructor entities will be replaced with native AutoCAD entities, which will be generated according to the display options provided. When all ShipConstructor entities have been successfully removed the drawing will be copied to the final location. The default name for the new file is the original file's name with '_Export' and, if necessary, a unique number appended to the end.

Layer Templates

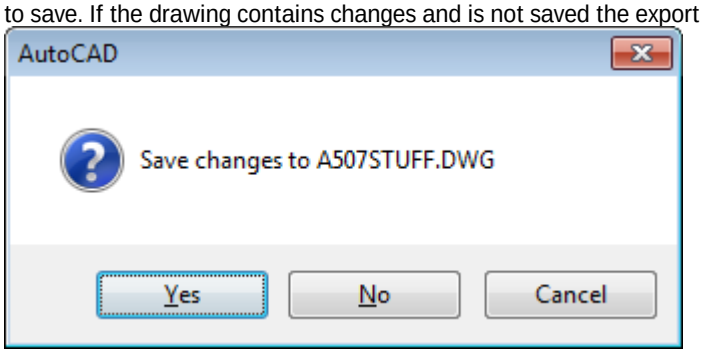
Layer templates are used to give fine grained control of where output entities are placed in the new drawing. The templates consist of static text that will be included as is in the layer name as well as any instance of the text '<Source Layer>' which will be replaced with the name of the layer the source entity is on. The line '<Source Layer>' can be included anywhere in the layer template and can be included multiple times.

If the new layer does not exist it will be created and the new entity or entities will be placed on it.

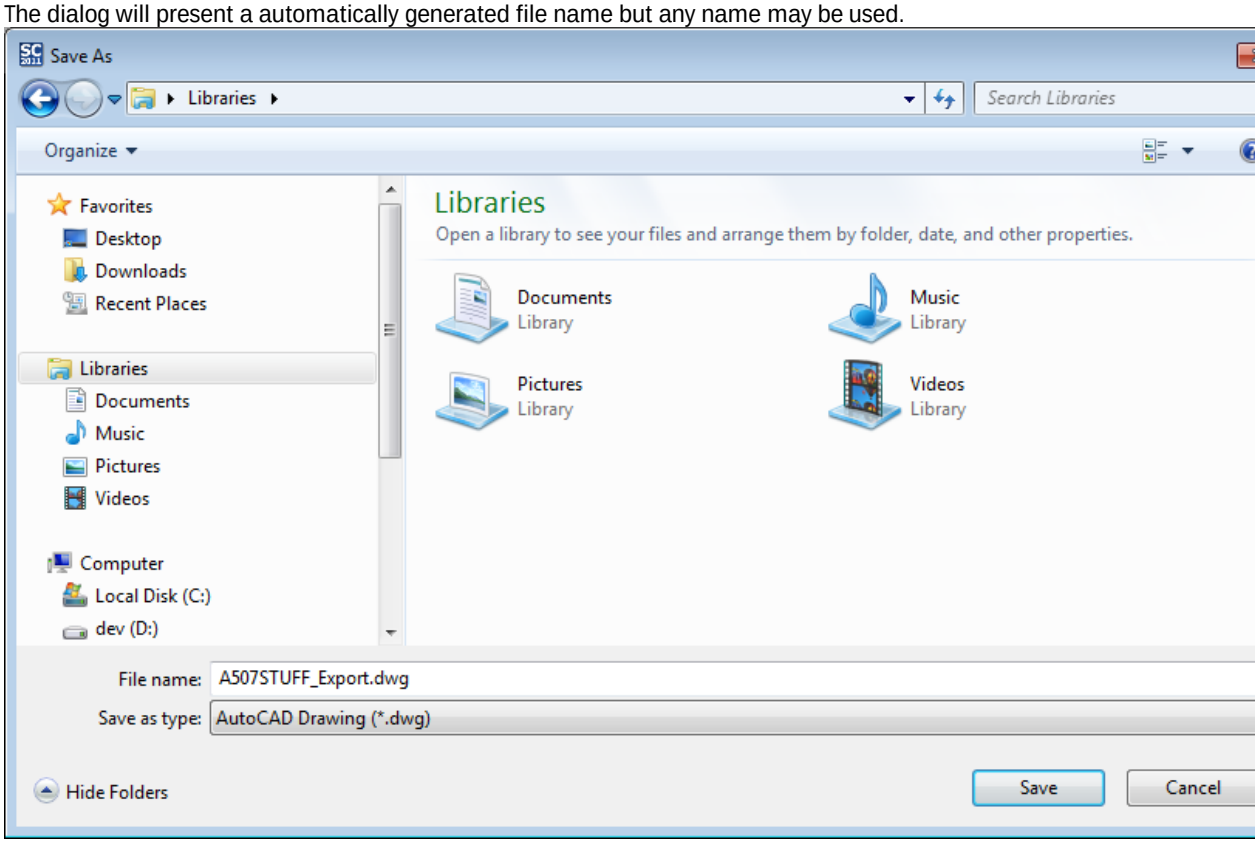
Source Layer	Layer Template	Final Layer
_Parts	ExportedParts	ExportedParts
_Parts	<Source Layer>	_Parts
_Parts	ExportLayer<Source Layer>	ExportLayer_Parts
_Parts	<SourceLayer>A<Source Layer>B	_PartsA_PartsB

Exporting using the Graphic User Interface

1) Choose SC Utilities > Export > Export to DWG (page 466)
2) If the current drawing has not been saved ShipConstructor will bring up a dialog box requesting permission to save. If the drawing contains changes and is not saved the export process will stop.



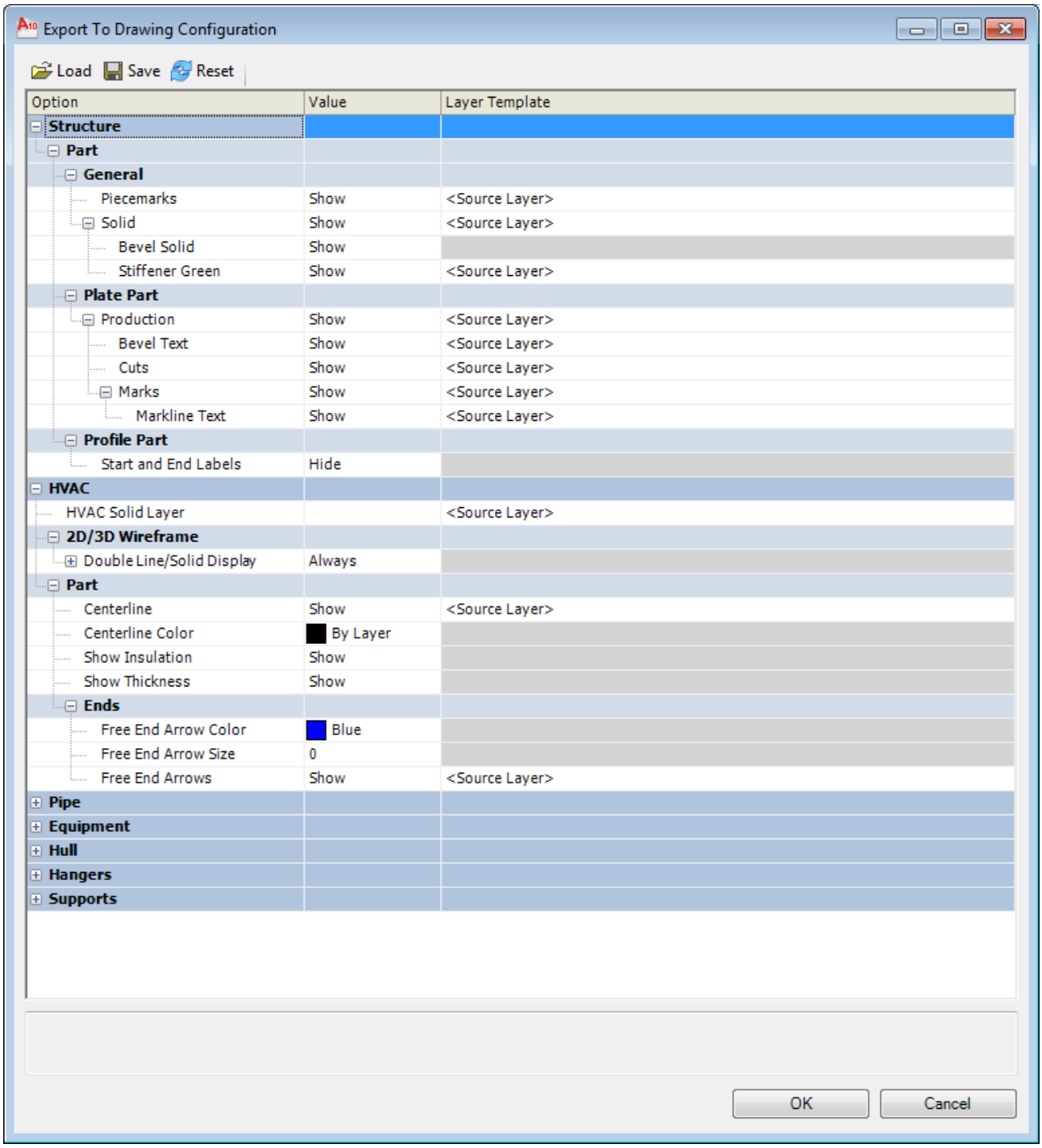
3) The Save As menu will appear allowing the user to select the name and location of the exported drawing. The dialog will present a automatically generated file name but any name may be used.



4) Once the configuration options have been properly set through the UI, or by loading a previous configuration click 'Ok' and the export will proceed.

   If you wish to save the configuration for use later, either with this UI or the silent export command press the 'Save' button and select an approprate file name.

   To load a previously saved configuration use the 'Load' button and select the configuration file to read.



Command Line Export

1) Enter the command -SCExportDwgSilent

2) The export will configure itself to use the ShipConstructor display settings of the current document, if you wish to use other settings entering 'Config' will allow you to enter an optional file name on the command line from which export settings can be read. These files can be generated using the user interface for SCExportDwg and the save option in the UI.

3) You will be prompted to enter a file name for the final exported document. ShipConstructor will automatically generate a file name which will be displayed on teh command line between the '<' and '>' charicters. If you press enter here that file will be used or a new file name can be entered on the command line.

4) When read pressing enter will start the export. You will be unable to stop the export once it has begun.

Fix Duplicate Drawing Names

By previous design, drawing name was not the primary idenfication in ShipConstructor database and the uniqueidentifier constraint was not set on the field. To get rid off some confusions and ambiguities, current version of ShipConstructor will bring in the constraint and provides a command to fix those duplicate drawing names existing in the system.

In the database updating process, Unique filename constraint was added in and all duplicated filenames in each unit were searched out and changed to new names by appending a string “$Duplicate$#” into their ends.

SCFIXDUPLICATEPRODDWGS command will go though all data records with special ending tags in database and check those related files in project folders. If a matching file was found, the filename in database will be recovered to the correct name; otherwise the record will be deleted from database.

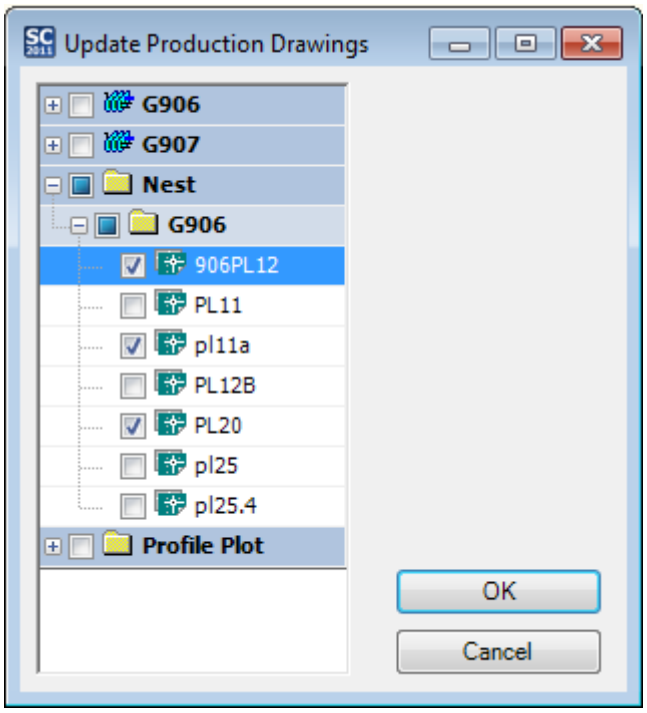
Command line Fix Duplicate

1) Enter the command SCFIXDUPLICATEPRODDWGS.
2) ShipConstructor will get rid off all duplicate filenames that don’t have a matching file in the folder and rename found ones back to original names. A report will be generated at the end telling which filenames were changed in the process.

Update Production Drawings

This command will allow the update of multiple production drawings at once. As this is a time consuming process the only user input required is the initial setup and once started the update will continue with out user input until complete.

1) Enter the command SCUPDATEPRODUCTIONDWGS
2) Select the drawings to update from the list.
    a. Check the “Ignore up to date drawings” to avoid calling update on any up to date drawings that are selected.



3) Click “Ok” to begin updating drawings. The updater will now open each drawing and call the silent update drawing command –SCUPDATEDWGSILENT. Once the process has completed alog file will be opened.

Appendix A: Menus, Tools & Commands

Manager Menus

File > Import Stocks >
Lets you import Manager settings from another project or from an XML file that was exported from another project, either for All Modules or for an individual menu in Manager (General, Structure, Pipe, HVAC, Equipment). See the Project Management manual for details.

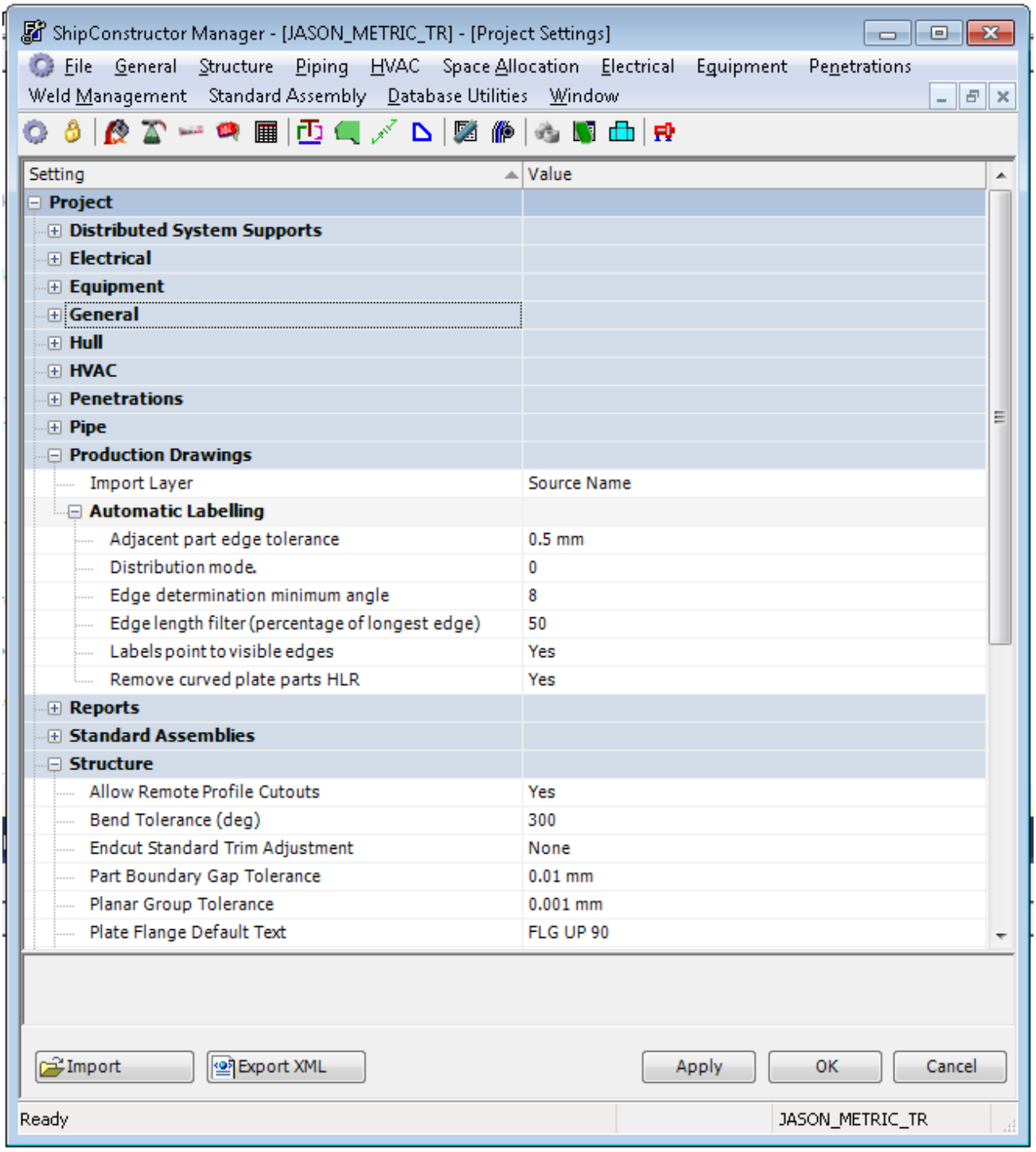
File > Export Stocks >
Exports Manager settings to an XML file, either for All Modules or for an individual menu in Manager (General, Structure, Pipe, HVAC, Equipment). See the Project Management manual for details.

File > Exit
Closes Manager. You are prompted to save any unsaved data.

General > Project Settings

Menu	General > Project Settings
Toolbar	
Navigator	Settings (with the project selected)
Permissions	None
Procedure	Project Settings (page 27)

Opens the Project Settings window, letting you set various project-wide settings.



Distributed System Supports

Electrical

Equipment

General

Administrator > Automatic Save – Specifies ShipConstructor's behavior when AutoCAD triggers an automatic save (auto-save).

Off – AutoCAD's auto-save has no effect on the ShipConstructor drawing or database.

OnUserControl – The ShipConstructor drawing and database are saved when the auto-save triggers.

OnResetOnStart – The ShipConstructor drawing and database are saved when the auto-save triggers. Additionally, each time you connect to a project, the AutoCAD auto-save settings are automatically set to be enabled (using a default time of 10 minutes between saves). After you have connected to a project you may turn off the auto-save in the AutoCAD options, but next time you connect it will be re-enabled.

Note: You may enable or disable AutoCAD's automatic save, and specify a time after which an auto-save will be triggered in the Open and Save tab of the AutoCAD Options window (Tools > Options menu).

Bill Of Materials > 'N/A' Text – When a field in the BOM table is not applicable for a part, the field contains the specified text.

Bill Of Materials > 'Varies' Text – When a row in the BOM table is a grouping of items and the data for the items is different for a field, the field contains the specified text.

Display Length Decimal Places – Indicates the precision of length values displayed in the user interface.

Display Length in ft in 1/16 format – Length measurements displayed in the user interface are displayed in a standard feet, inches, and inch-fractions format or in decimal format.

Display Length Units – Indicates the units of length values displayed in the user interface.

Display Weight Decimal Places – Indicates the precision of weight values displayed in the user interface.

Display Weight Units – Indicates the units of weight values displayed in the user interface.

Minimum Interference Volume – The minimum volume that will result in an interference. This lets you filter out very small interferences during interference checking that may be the result of parts just touching, interpolation errors by the computer, or interferences so small that production will not notice them.

Project ID – The unique project identifier.

Project Length Units – The length units used throughout the project (for example, the units that AutoCAD geometry will be drafted in). Do not change this setting after you have begun working on a project.

View From Aft – This setting indicates which direction modeling in Frame Planar Group Model Drawings will take place. This setting should be on if you prefer looking at frames from aft to forward. This setting is determined when the project is create and is not changeable.

Hull

HVAC

HVAC Colors

HVAC Sizes

Penetrations

Pipe

Pipe Colors

Pipe Sizes

Production Drawings

Import Layer – The default setting for part layers when creating a production drawing. Source Name – uses the name of the source for the layer name; Retain Model Drawing Layers – Places parts on the same layer as they are in the model drawings.|keyword=Import Layer

Automatic Labeling

Adjacent part edge tolerance - Edges will be less likely to be labeled if they are closer than this distance to an edge of another part. Default value is 0.5 mm. See Adjacent Part Edge Tolerance (page 249)

Distribution mode - Automatic distributing functions will use this setting as the distribution mode. User can set the value to one of the three distribution modes – Nearest(0), Center(1) and Equidistant(2). Default Value is 0 (Nearest).

Edge determination minimum angle - Line segments will be considered part of the same edge if they are less than this angle away from tangent. Default Value is 8 degrees. See Edge Determination Minimum Angle (page 251)

Edge length filter (percentage of longest edge) - Edges shorter than this percentage will be filtered out from consideration of being pointed to by the label. Default Value is 50 %. See Edge Length Filter (page 250)

Label point to visible edges - Turns labels automatic label generation from label on piecemark to label on visible part edge. Default Value is No. See Label on Visible Edge On/Off (page 249)

Remove curved plate parts HLR - Turns on option to include curved plates in hidden line calculation when label on visible part edge is on. Default Value is No. See Curved Plates in Visible Edge Detection On/Off (page 250)

Reports

Company Name – A text field that can be inserted into production drawings as a keyword. This is included in the header of reports.

Hull Number – A text field that can be inserted into production drawings as a keyword.

Job Number – A text field that can be inserted into production drawings as a keyword.

Project Description – A text field that can be inserted into production drawings as a keyword.

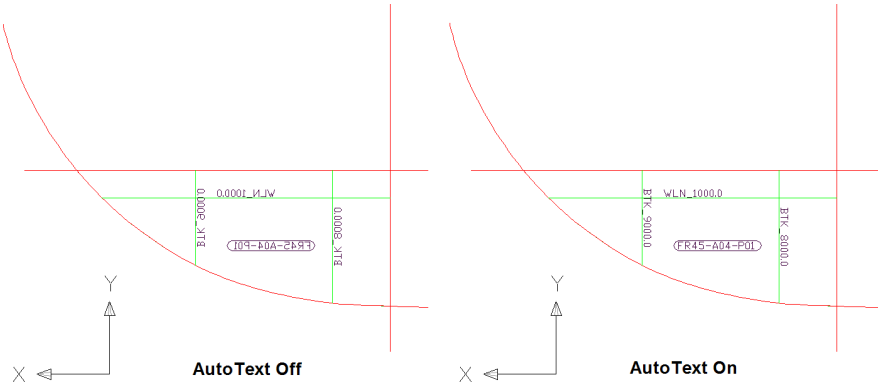
Project Name – The descriptive name of the project that will appear in report headers and title pages.

Standard Assemblies

Structure

Allow Remote Profile Cutouts – Include cutouts from Profiles in different units when adding cutouts to a plate.

AutoOrient Text – When looking at text in structure parts, the text can be viewed as being always readable no matter your viewpoint. This is not the default AutoCAD behavior. You may be more used to the AutoCAD behavior to verify you are looking from the back side.



The difference on the text when viewing from the back side

AutoOrient Text

3D PDM – This setting affects the text while in any product model drawing.

Arrangement Drawings – This setting affects the text while in pipe, HVAC or equipment arrangement drawings.

Assembly Drawings – This setting affects the text while in assembly drawings.

Nest Drawings – This is always on because marked text should always be readable.

Spool Drawings – This setting affects the text while in pipe or HVAC spool drawings.

Bend Tolerance (deg) – The minimum angle (in degrees) at which a curved faceplate is considered to have a bend. A curved faceplate following the hull is usually represented by a polyline with small straight-line segments. The directional change from segment to segment is usually very small (in the order of 1/10 of a degree). Bends this small are not producible in a shipyard environment. However, over the length of the faceplate, the total directional change is quite likely very large. The Bend Tolerance lets you set a threshold above which the bend will be listed in the bend report. When the accumulated values of directional change of the polyline segments exceed the Bend Tolerance value, the directional change is listed. Changing this value after faceplates have been made does not affect their bend data.

NC-Pyros Export – See Export To NC-Pyros Window (page 523).

Planar Group Tolerance – The maximum distance that planes defined by additional geometry selected when creating a planar group can be offset from the plane that is used to define the planar group. Sometimes geometry imported into AutoCAD from other software is not exactly in one plane.

Plate Flange Default Text – The default text to use on the label of a plate flange.

Plate Nesting

Display FS Marking as Dashed – Yes – Far side marks (the bottom side as it is shown in the nest drawing) is shown using the SConDashed Linetype style: No – Far side marks are shown as they will be actually marked.

Nest Corner – See Autonesting Options

Nest Fit Method – See Autonesting Options

Nest Rotation Angle – See Autonesting Options

Nest Vertically – See Autonesting Options

Solid Creation Tolerance – Determines how accurate ShipConstructor solids are. Solids are used for visual representation, interference checking, and the calculation of weight and center of gravity. Solids are not used for production. Set Solid Creation Tolerance to 0 to ensure solids exactly match the production geometry or increase the value to decrease the accuracy of solids and reduce file size.

Stiffener Marking Type –Controls how ShipConstructor marks stiffeners on plates. A setting of Extents marks the full length of the stiffener while a setting of Contact marks only the length of the stiffener which contacts the plate.

The difference between Contact setting and Extents setting.

Structure Colors

Construction Line Colors

Offset Construction Line Color – The color of all Offset construction lines.

Planar Group Hull Trace Color – The color of all HullTrace construction lines.

Planar Group Plane Color – The color of all PlanarGroupPlane construction lines.

User Color – The color of all User construction lines.

Draft Colors

Draft Cut Color – The color assigned to the _Draft_Cut layer in structure model drawings.

Draft Mark Color – The color assigned to the _Draft_Mark layer in structure model drawings.

Draft NoProcess Color – The color assigned to the _Draft_NoProcess layer in structure model drawings.

NC-Pyros Colors – The colors of parts when displayed in a nest drawing or after they have been assigned to a nest. This makes it easy to differentiate between parts that have been assigned to a nest and parts that are in a nest drawing but have not yet been nested.

Bevel Angle Color – Color of the Bevel angle text

Bevel Mark Color– Color of the Bevel mark text

Bevel Standard Color– Color of the Bevel standard text

Inset Color – Outside cut that is in a hole of another part.

Inside Color – Inside cuts.

Mark Color – Marking on the plate.

NoProcess Color – Text and lines that are there for reference but not marked on the plate.

Outside Color – Outside cuts.

Plate Stock Color – Stock plate border.

Remnant Colors – Colors of the remnant components in the DXF file after being exported

Remnant Cutline Color – Remnant cutline.

Remnant Hatch Color – Remnant hatch.

Remnant Text Color – Remnant text.

Nest Part Colors

Outdated Part Color – Part that has been modified after it has been nested.

Unassigned Part Cut Color – A temporary color to signify that the part is not assigned to a nest.

Unassigned Part Mark Color – Temporary color to signify that the part is not assigned to a nest.

Unassigned Part NoProcess Color – Temporary color to signify that the part is not assigned to a nest.

Part Invalid Color – When a plate part cannot generate a boundary, the part will change to this color. It is good to make this a distinctive color that will stand out clearly.

Plate Part Colors

Part Bevel Color – Color of bevel code text.

Part Cut Color – Color of lines that will be cut by the NC machine.

Part Mark Color – Color of lines that will be marked by the NC machine.

Part NoProcess Color – Color of lines that will not be processed by the NC machine.

Profile Part Colors

Part Green Color – When adding Green to a stiffener or faceplate, the green is represented by a solid using the Part Green Color.

Toolpath Gap Tolerance – The toolpath command will connect lines that are not connected by a distance. The largest gap that the toolpath command will connect is this gap tolerance.

Twisted Stiffener Import Units – The units of STR stringer files from ShipCAM or Maxsurf when creating new twisted stiffeners.

Use Surface Name for Part Name – Creating curved plate parts can be named from the surface name in the Hull drawing or the DXF filename if using Curved Plates > New.... If this is set to No, a new autonumbered name is generated.

Weld Shrinkage Icon Size – The default size of the weld shrinkage icon.

Weld

General > User Permissions

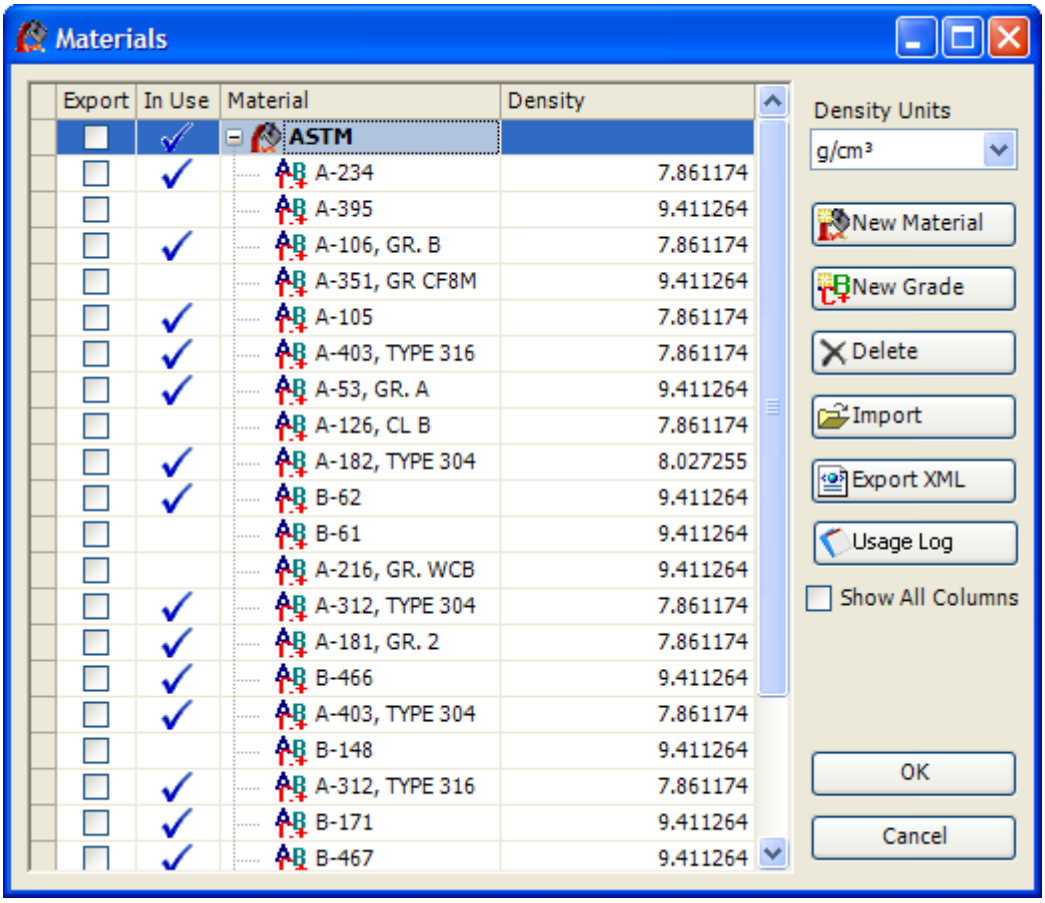
Menu ..... General > User Permissions
Toolbar.....
Permissions ... General > Edit User Permissions
Procedure..... User Permissions (page 28)

Opens the User Permissions window, letting you set up user permissions and permission groups. See User Permissions in the Project Management manual for details.

General > Materials

Menu ..... General > Materials
Toolbar.....
Permissions ... General > Manager > Edit Materials
Procedure..... Materials (page 29)

Opens the Materials window, letting you create and edit materials.



Export– Checked materials and grades are exported when you click Export XML.

Material – Lists materials and grades. Right-click on a material or grade to Rename it, Delete it, or create a New Material or New Grade.

Density – The density of the material, measured in the Density Units. Before entering a Density value, set the Density Units to the units for the value you want to enter.

In Use – Indicates with a checkmark which materials and grades are currently in use in the project.

Density Units – The Density of materials and grades are measured in the Density Units. Before entering a Density value, set the Density Units to the units for the value you want to enter. To display material densities in another unit, change the Density Units.

New Material – Creates a new material.

New Grade – Creates a new material grade for the selected material.

Delete – Deletes the selected material or grade.

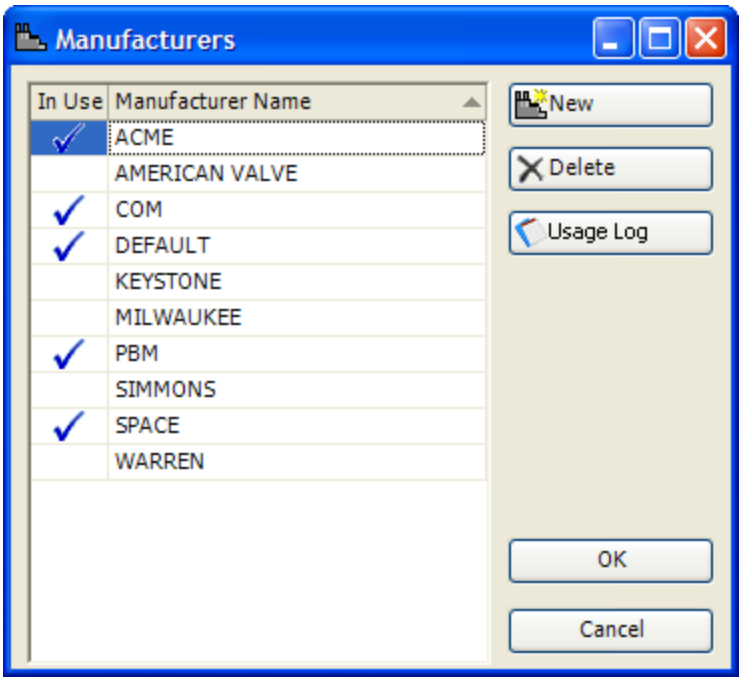
Import – Lets you import materials and grades from another project or from an XML file that was exported from another project.

Export XML – Exports the checked Export materials and grades to an XML file.

General > Manufacturers

Menu ..... General > Manufacturers
Toolbar..... None
Permissions ... General > Manager > Edit Manufacturers
Procedure..... Manufacturers (page 32)

Opens the Manufacturers window, letting you create and edit manufacturers.



In Use – Indicates with a checkmark which manufacturers are currently in use in the project.

Manufacturer Name – Lists manufacturers.

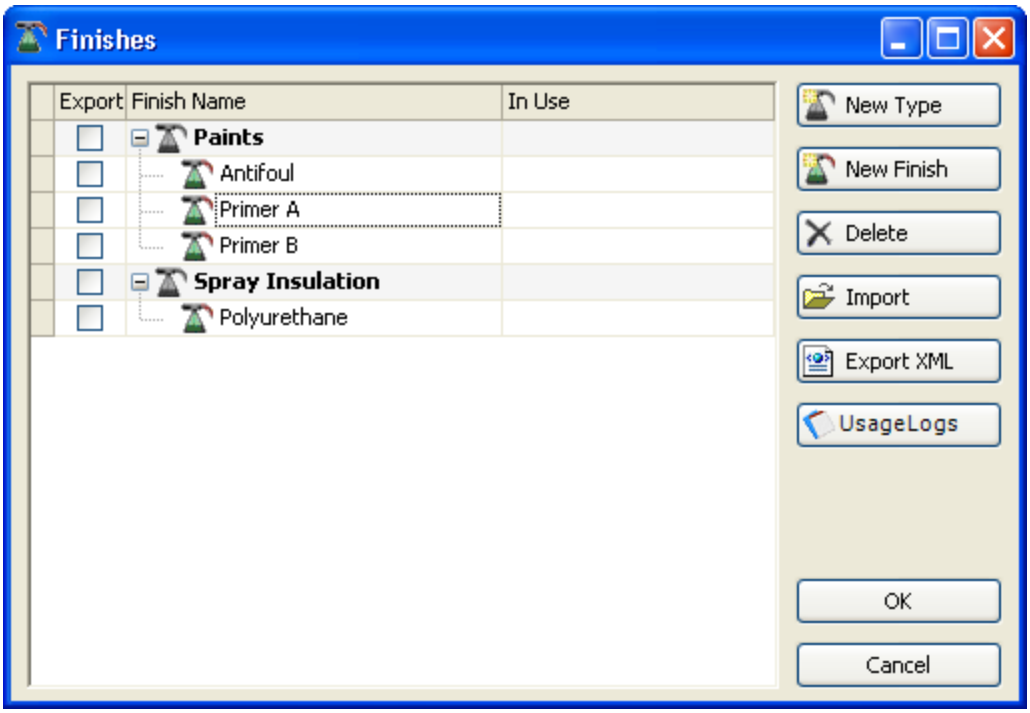
New – Creates a new manufacturer.

Delete – Deletes the selected manufacturer.

General > Finishes

Menu ..... General > Finishes
Toolbar.....
Permissions ... General > Manager > Edit Finishes
Procedure..... Finishes (page 33)

Opens the Finishes window, letting you create and edit finishes.



Export – Checked finishes and finish types will be exported when you click Export XML.

Finish Name – Lists finishes and finish types. Right-click on a finish or finish type to Rename it, Delete it, or create a New Finish or New Finish Type.

In Use – Indicates with a checkmark which finishes and finish types are currently in use in the project.

New Type – Creates a new finish type.

New Finish – Creates a new finish for the selected finish type.

Delete – Deletes the selected finish or finish type.

Import – Lets you import finishes and finish types from another project, from an XML file that was exported from another project or from a Common Parts Catalog (CPC) bulk import file.

Export XML – Exports the checked Export finishes and finish types to an XML file.

Usage Logs – Generates a list of all parts using the selected finish.

General > Insulation

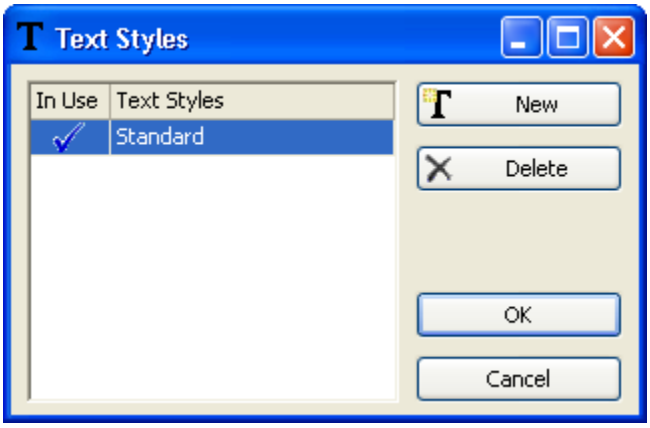
Menu ..... General > Insulation
Toolbar..... None
Permissions ... General > Manager > Edit Insulation
Procedure..... See the HVAC manual

Opens the Insulation window, letting you create and edit insulation. See Insulation in the HVAC manual for details.

General > Text Styles

Menu ..... General > Text Styles
Toolbar..... None
Permissions ... General > Manager > Edit Text Styles
Procedure..... Text Styles (page 35)

Opens the Text Styles window, letting you create and edit text styles.



General > Accessory Packages

Menu ..... General > Accessory Packages
Toolbar..... None
Permissions ... General > Manager > Edit Accessory Packages
Procedure..... See Pipe manual

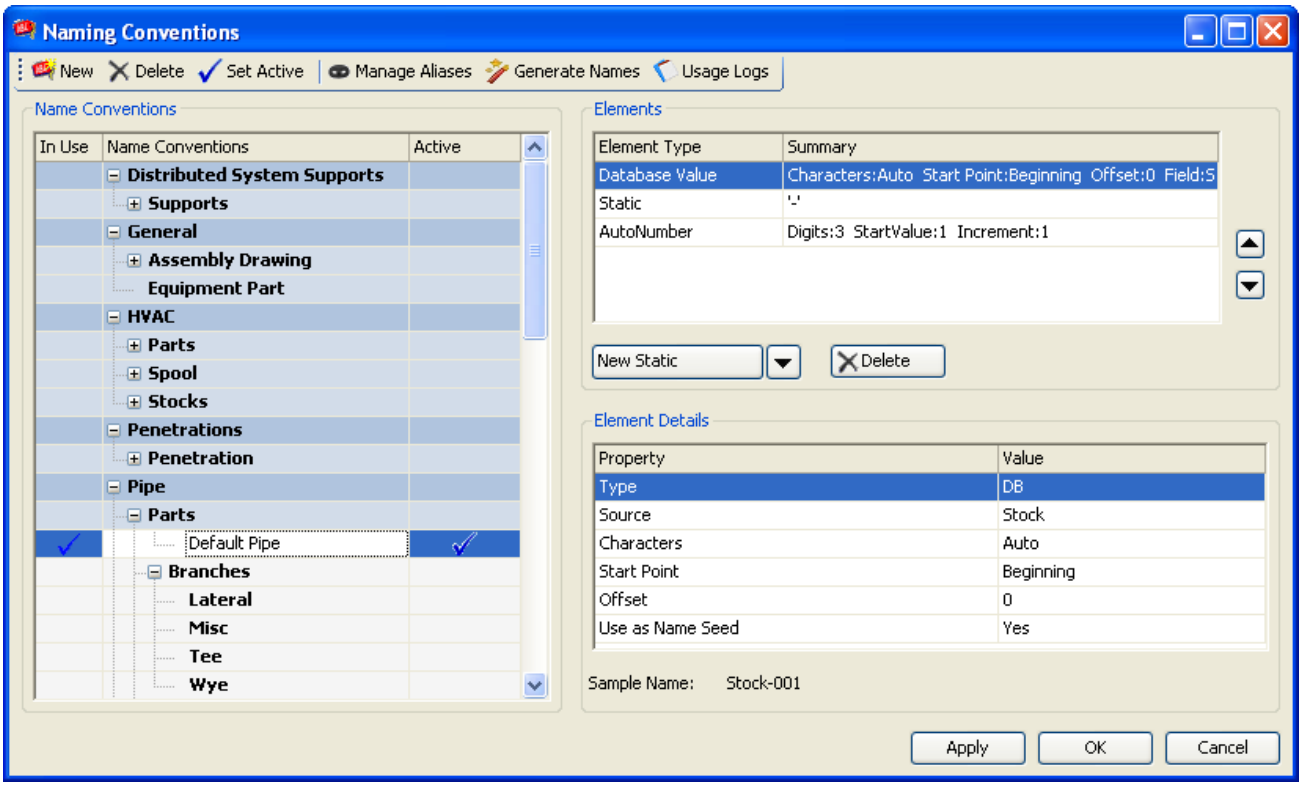
Opens the Accessory Packages window, letting you create and edit accessory packages. See Accessory Packages in the Pipe manual for details.

General > Naming Conventions

Menu ..... General > Naming Conventions
Toolbar..... None
Permissions ... Structure > Manager > Edit Naming Conventions
Permissions ... Pipe > Manager > Edit Naming Conventions
Permissions ... HVAC > Manager > Edit Naming Conventions
Permissions ... Penetrations > Manager > Edit Naming Conventions
Permissions ... General > Manager > Edit Assembly Drawing Naming Conventions
Procedure..... Naming Conventions (page 35)

Opens the Naming Conventions window, letting you create and edit naming conventions.

Naming Conventions Window



Name Conventions – Lists all naming conventions, grouped by type. Active naming conventions are indicated by a checkmark. Select a naming convention to display its Elements.

New – Creates a new naming convention within the selected group.

Delete – Deletes the selected naming convention.

Set Active – Makes the selected naming convention the active naming convention for its group.

Manage Aliases – Opens the Alias Manager, letting you change the the names of sub-categories that are used to organize naming conventions.

Elements – Lists the elements (the components that make up a naming convention) of the selected naming convention in order. Select an element to display its properties below.

Up/Down Arrows – Moves the selected element up or down in the list of elements, letting you re-order them.

New – Opens the Select Type window, letting you add an element to the selected naming convention.

Delete – Deletes the selected element from the selected naming convention.

Property, Value – Displays the properties and values of the selected element and lets you edit the values.

Understanding Seeding

Seeding gives you more control over how your names are generated. This is best illustrated by an example. Suppose you have a naming convention defined with three elements: Material-System-Auto-number with the auto-number starting at 1 and the increment being 1. The first part you create may have a name like this:

M1-FreshWater-001

In this example, the part is made from a material named M1 and is in the Fresh Water system. For this example we will assume that both the Material element and the System element are seeds. If you create a new part that was also made from material M1 and in system Fresh Water, the name will be as follows:

M1-FreshWater-002

If you create a third part that was made of material M1 but was in the Salt Water system, ShipConstructor will generate a part name as follows:

M1-SaltWater-001

Notice how the number has restarted at 1. This is because one of your seed elements has changed, in this case the System. If you create a fourth part, this time back in the Fresh Water system but with material M2, ShipConstructor would generate a name as follows:

M2-FreshWater-001

Again the numbering scheme has changed because the combination of your two seed elements does not match anything we have seen previously. If you now create a fifth and sixth part made from material M2 and in system Fresh Water, ShipConstructor will generate the names as follows:

M2-FreshWater-002

M2-FreshWater-003

To further illustrate the nature of seeding, consider the following example. To make things simple, pretend that you are again working with a fresh project and have not generated the names above. You define a name convention identical to the one in the example above (Material-System-Auto-number) except that instead of both the Material and System elements being seeds, only the Material is a seed element. Following the same procedure as above your part names will be as follows:

M1-FreshWater-001

M1-FreshWater-002

M1-SaltWater-003

Notice how even though the system changed from Fresh Water to Salt Water, the numbering scheme did not change; it continued counting up. This is because the system is not a seed element. If you create two more parts, the names will be as follows:

M2-FreshWater-001

M2-SaltWater-002

Notice how the numbering scheme changed the first time because the material element changed but continued counting up the second time.

Database Elements

The power in naming conventions lies in the ability to generate names based on entity attributes stored in the database. The following are descriptions of all the database elements currently supported by ShipConstructor.

Assembly returns the primary product hierarchy assembly name of an ancestor assembly of your entity. Which level's assembly name is returned depends on which Assembly Tree Level you select from the Tree Level drop-down list. The list of assembly levels depends on how you structure your Product Hierarchy tree.

> Note: If the part is in an assembly with a level higher than the selected assembly tree level, nothing is returned for that element.

Extrusion Profile Standard Type applies to stiffeners. Examples of extrusion profile standard types are angle, bulb flat, structural pipe, custom, and tee. Extrusion profile standard types can be aliased using the Naming Convention Alias Manager.

File Path applies to Plate Nests. It returns the path to the plate nest drawing starting from the Nest folder. The purpose of the file path element is to let you put the product hierarchy location in the plate nest name.

This is done by creating a directory structure in the Nests folder that matches the product hierarchy and then putting plate nest drawings at the appropriate location in the folder structure.

Major Stock Type applies to all Structure parts. Examples of major stock types are plate, extrusion, corrugated and plank. Major stock types can be aliased using the Naming Convention Alias Manager.

Material returns the material name of the stock.

Material Grade returns the material grade of the stock.

Nest Drawing returns the nest drawing name of plate nests.

Nominal Size returns the nominal size of the pipe stock.

Part Side returns the side of the ship on which the structure part lies.

Part Type applies to Structure parts. Examples of part types are plate, stiffener, corrugation, faceplate, and plank. Part types can be aliases using the Naming Convention Alias Manager.

Planar Group Model Drawing returns the planar group model drawing that the structure part is modeled in.

Plate Stock Length returns the length of the plate stock item that the structure part is made from.

Plate Stock Width returns the width of the plate stock item that the structure part is made from.

Pressure Rating returns the pressure rating of the pipe stock.

Profile Plot Drawing returns the profile plot drawing name of profile nests.

Project returns the name of the ShipConstructor project.

Remnant returns the name of the remnant that the plate nest was cut from.

Sheet Stock returns the name of the sheet stock that the HVAC stock is made from.

Stock returns the name of the stock that the part is made from.

System returns the spec level, system level, or branch level system name of the system of your entity. Which level is returned depends on your selection in the Tree Level drop-down list.

Note: If the entity is in a system with a level higher than the selected system tree level, nothing is returned for the System of that element.

Thickness returns the thickness of the plate stock that the plate part is made from.

General > Production Output > Label Styles

Menu ..... General > Production Output > Label Styles
Toolbar..... None
Permissions ... General > Manager > Edit Label Styles
Procedure..... Label Styles (page 44)

Label styles are used to define annotations in production drawings.

General > Production Output > Dimension Styles

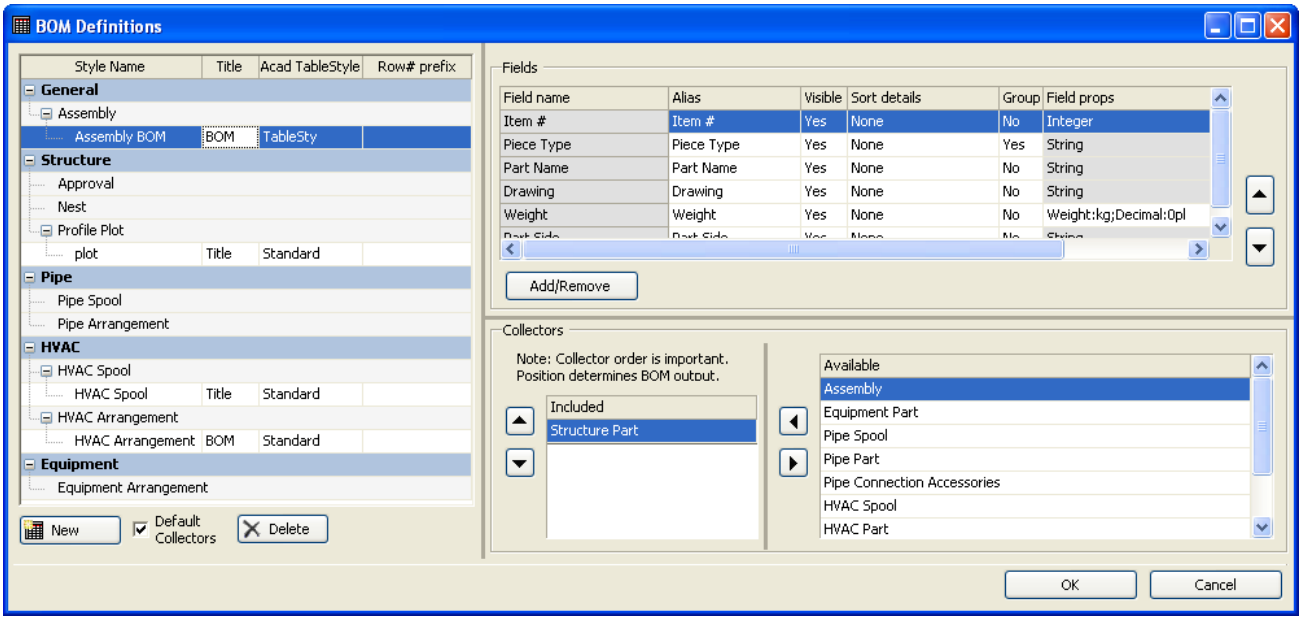
Menu ..... General > Production Output > Dimension Styles
Toolbar..... None
Permissions ... General > Manager > Edit Dimension Styles
Procedure..... Dimension Styles (page 45)

General > Production Output > Bill Of Materials

Menu ..... General > Production Output > Bill Of Materials
Toolbar..... None
Permissions ... General > Manager > Edit Bill Of Materials
Procedure..... Bill Of Materials (page 45)

ShipConstructor integrates all Bill of Materials management through the BOM Definitions manager. The BOM Definitions manager lets you create and edit existing BOMs for all production drawing types.

Access the BOM Definitions manager through Manager > General > Production Output > Bill of Materials.



BOM Definitions display
Broken down into production drawing categories, the BOM Definitions area lets you create and delete definitions and editing characteristics specific to a BOM.

- Style Name – Name of the BOM definition.
- Title – Title to display on AutoCAD tables.

  Note: Only tables using table styles that contain the Title section will display the Title text.

- Acad TableStyle – AutoCAD tablestyle to use for BOM instances. If the specified tablestyle is not found at the time of insertion, then the STANDARD style is used.
- Label Style – ShipConstructor label style to use for the BOM. This style can be overridden by the Collector Label Style.
- Row # prefix text – Precedes item number with text in both table and label.

New
Creates new BOM definitions.

Default Collectors – If selected, the BOM definition includes reasonable collectors for the current production drawing type. You can modify the collectors and their order after creation.

Delete
Deletes the selected BOM definition. If the BOM definition is in use, it cannot be deleted.

Moves selected fields up in the listing order

Moves selected fields down in the listing order.

Fields
Lets you customize aspects of the fields that appear in your BOM:

- Choose which fields to include in the BOM.
- Alias the field names to something appropriate for your company.
- List the order the fields will be displayed in.
- Specify sorting, grouping, and visibility of the included fields.
- Specify properties specific to the type of field included, be it a decimal, length, weight, or other type of value. Fields with manageable properties are indicated by a white background in the Field Props column.

- Add/Remove – Adds or removes production fields or user attribute fields. Production fields are those provided by ShipConstructor and that exist in every project. Attribute fields are special fields that are defined by the user for the project. All attribute fields are listed, regardless of whether they logically make sense for the BOM definition.

Collectors

Collectors are responsible for gathering objects in a drawing to be included in a BOM and then processing the object data for display in the BOM.

Note: Collector order is important because each collector returns information specific to the objects it collects. Resultant information for each collector is then compiled in the order specified by the BOM definition.

Collectors are specific for each production drawing type. The following table lists the collectors available per drawing type.

Module	Production Drawing Type	Collectors
General	Assembly	Assembly, Structure Part, Equipment Part, Pipe Spool, Pipe Part, Pipe Connection Accessories, HVAC Spool, HVAC Part, HVAC Connection Accessories, Penetration Part, General Part
Structure	Approval	To be determined
	Nest	Plate Nest
	Profile Plot	Profile Plot
Pipe	Pipe Spool	Pipe Part, Pipe Connection Accessories, Pipe Cutlist
	Pipe Arrangement	Assembly, Structure Part, Equipment Part, Pipe Spool, Pipe Part, Pipe Connection Accessories, HVAC Spool, HVAC Part, HVAC Connection Accessories, Penetration Part, General Part
HVAC	HVAC Spool	HVAC Part, HVAC Connection Accessories, HVAC Cutlist
	HVAC Arrangement	Assembly, Structure Part, Equipment Part, Pipe Spool, Pipe Part, Pipe Connection Accessories, HVAC Spool, HVAC Part, HVAC Connection Accessories, Penetration Part, General Part
Equipment	Equipment Arrangement	Assembly, Structure Part, Equipment Part, Pipe Spool, Pipe Part, Pipe Connection Accessories, HVAC Spool, HVAC Part, HVAC Connection Accessories, Penetration Part, General Part

Collector Functionality

The following table lists the collectors and their functions

Collector	Description	Additional Functionality
Assembly	Organizes objects into primary production strategy assemblies. Returns information for each of the first level subassemblies. Main assembly and children of the subassemblies are not reported.	
Structure Part	Finds and returns information for all structure parts.	
Equipment Part	Finds and returns information for all equipment parts.	
Pipe Part	Finds and returns information for all pipe parts.	Lets you specify the type and spool status of the objects listed in the BOM.
Pipe Connection Accessories	Queries all pipe-pipe connections and retrieves accessory package information from them.	
Pipe Spool	Gathers all Pipe objects and organizes them into their spools. Returns information for each spool.	
HVAC Part	Finds and returns information for all HVAC parts.	Lets you specify the type and spool status of the objects listed in the BOM.

HVAC Connection Accessories	Queries all HVAC- HVAC connections and retrieves accessory package information from them.	
HVAC Spool	Gathers all HVAC objects and organizes them into their spools. Returns information for each spool.	
Penetration Part	Gathers all Penetration parts and reports information for the items contained in each.	
General Part	Collects all parts and returns part level information for each. Warning: The general part collector can take a significant amount of time to process data on larger projects.	
Nest	Finds and returns all objects in a Nest drawing specific to a particular nest.	
Profile Plot	Collects all objects in a Profile Plot drawing specific to a particular nest.	

General > Production Output > Predefined Assembly Format

Menu ..... General > Production Output > Predefined Assembly Format
Toolbar..... None
Permissions ... General > Manager > Edit Predefined Assembly Format
Procedure..... Predefined Assembly Format (page 49)

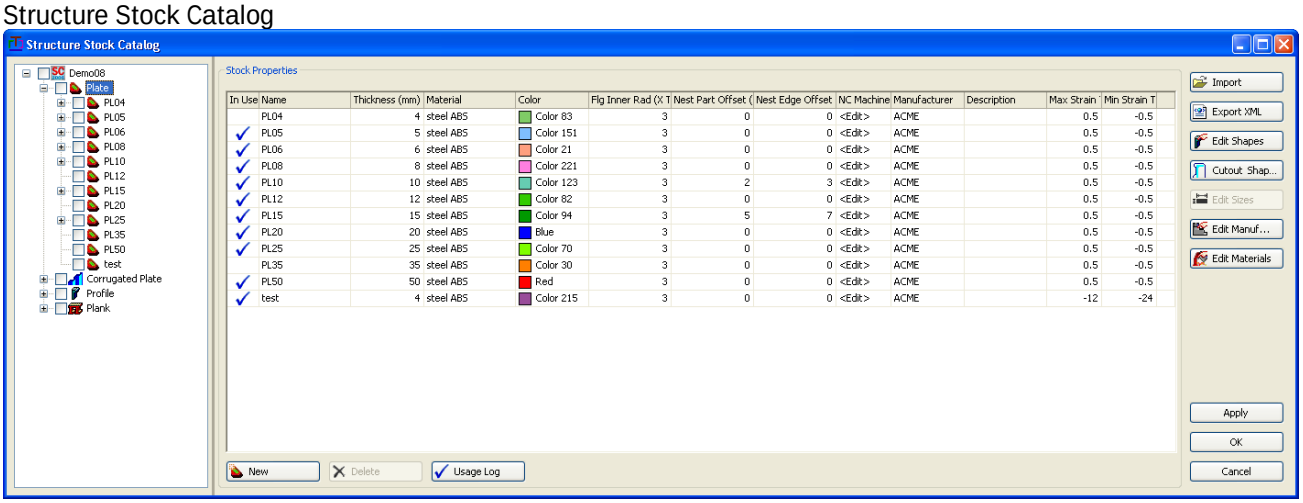
Structure Menu

Structure > Stock Catalog

Menu ..... Structure > Stock Catalog
Toolbar.....
Permissions ... Structure > Manager > Edit Structural Stocks
Procedure..... Stock Catalog (page 51)

Opens the Structure Stock Catalog window and lets you create and edit structure stocks.

Structure Stock Catalog



Structure Stock List – Lists all structure stocks by type. Select a stock type to list stocks of that type under

Stock Properties – Lists stock properties for the stock or stock type selected in the structure stock list. See these sections for further information on individual properties:

- Plate Stock Properties (page 52)
- Profile Stock Properties (page 53)
- Corrugated Plate Stock Properties (page 52)

New – Creates a new stock of the type selected in the Structure Stock List.

Delete – Deletes the stock selected under Stock Properties.

Usage Log – Generates a list of all parts using the stock selected under Stock Properties.

Import – Lets you import stocks from another project, from an XML file that was exported from another project or from a Common Parts Catalog (CPC) bulk import file.

Export XML – Exports the checked stocks to an XML file.

Edit Shapes – Opens the Profile Shapes window, letting you create and edit profile stock shapes.

Cutout Shap... – Opens the Cutouts window, letting you create and edit cutouts. Cutouts can be inserted into plate or stiffener parts.

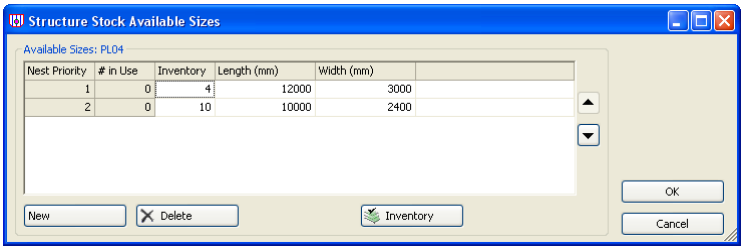
Edit Sizes – Opens the Structure Stock Available Sizes window, letting you specify the available sizes of a stock. See Edit Sizes Window (page 343).

Edit Manuf... – Opens the Manufacturers window. See General > Manufacturers (page 333).

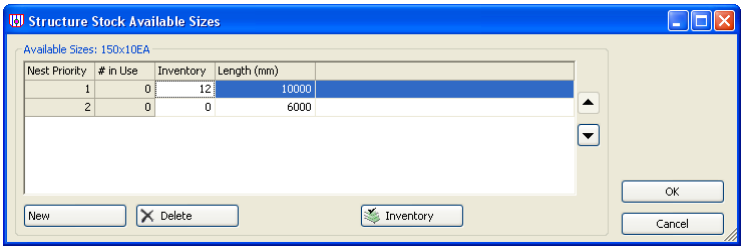
Edit Materials – Opens the Materials window. See General > Materials (page 332).

Edit Sizes Window

Stocks can come in various stock sizes. The edit sizes window is where you enter this information. You also track how many you have in inventory.



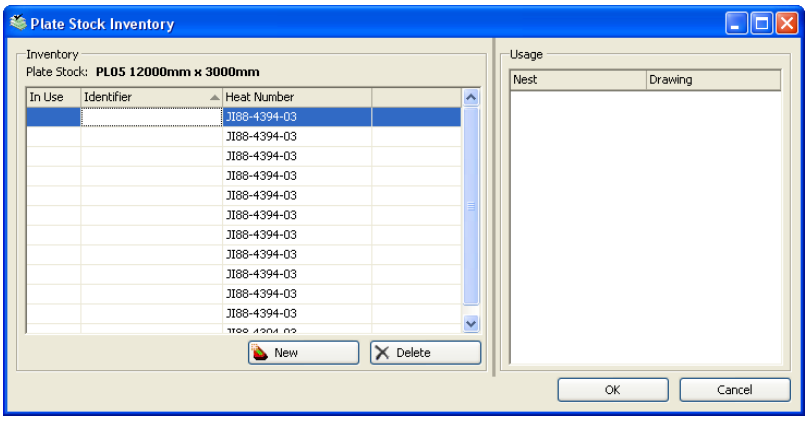
Available Sizes for a Plate Stock



Available Sizes for a Stiffener Stock

- Nest Priority – When multiple size stock sheets are defined automatic nesting must choose one to nest on. Nest Priority determines which size to try first.
- # In Use – The number of plates that are in nest drawings.
- Inventory – The total number of plates of the specified size in the project.
- Length – Length of the stock plate (profile stock)
- Width – Width of the stock plate (plates only)
- New – Create a new stock size.
- Delete – Delete the selected stock size. You cannot delete a stock size that is currently in use.
- Inventory – Shows the Inventory window. See Inventory Window (page 344).

Inventory Window



Inventory – The list of pieces for the selected size as indicated.

Identifier – The user-defined identifier for that piece of stock. Each piece can optionally have a different identifier.

Heat Number – The heat number tagged on the piece. Typically the heat number is entered in the Nest Manager after the nest is cut, but you can also enter it here.

In Use – Indicates if the stock item is in use in a nest.

Structure > Structure Stock Catalog > Import
Allows you to import stocks from an XML file.

Structure > Structure Stock Catalog > Export XML
Exports the checked stocks to an XML file.

Structure > Structure Stock Catalog > Edit Shapes
Edits the stiffener cross sections.

Structure > Structure Stock Catalog > Cutout Shap...
Edits the cutout types and cutout shapes.

Structure > Structure Stock Catalog > Edit Sizes
Edits the available sizes and quantities of the selected stock.

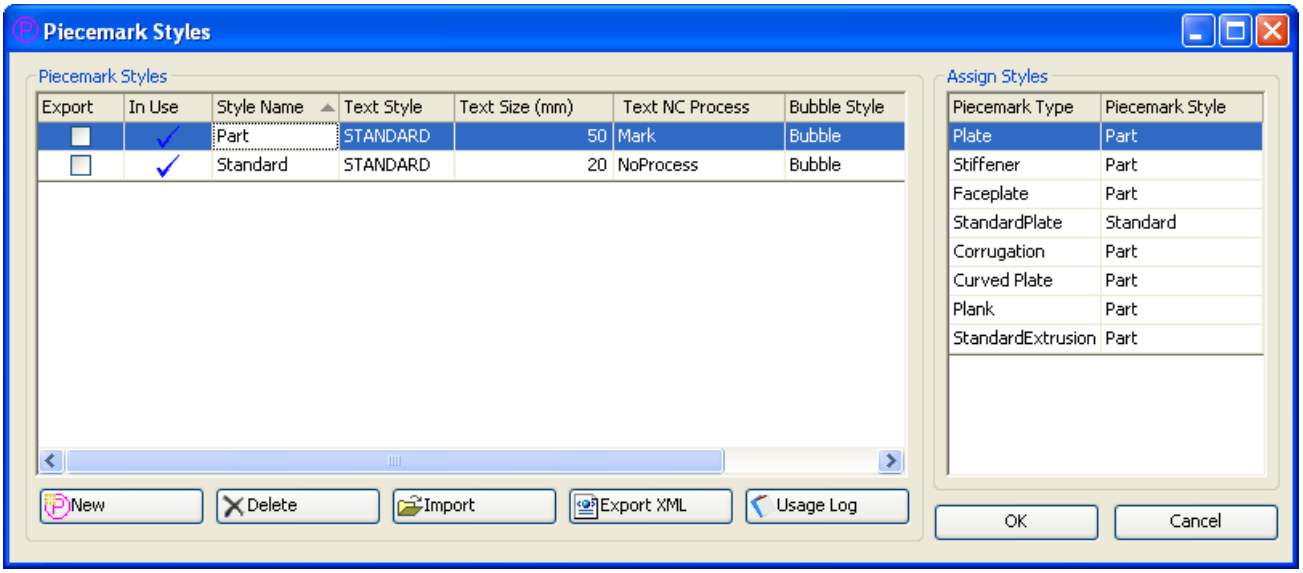
Structure > Structure Stock Catalog > Edit Manuf...
Allow you to edit and add manufacturers without closing the Structure Stock Catalog window. See General > Manufacturers (page 333).

Structure > Stock Stock Catalog > Edit Materials
Allow you to edit and add materials without closing the Structure Stock Catalog window. See General > Materials (page 332).

Structure > Piecemark Styles

Menu ..... Structure > Piecemark Styles
Toolbar..... None
Permissions ... Structure > Manager > Edit Piecemark Styles
Procedure..... Piecemark Styles (page 70)

Opens the Piecemark Styles window, letting you create and edit piecemark styles.



Piecemark Styles – Lists all piecemark styles and their properties.

New – Creates a new piecemark style.

Delete – Deletes the selected piecemark style.

Assign Styles – Lets you select a Piecemark Style for each Piecemark Type.

Structure > User-Defined Attributes

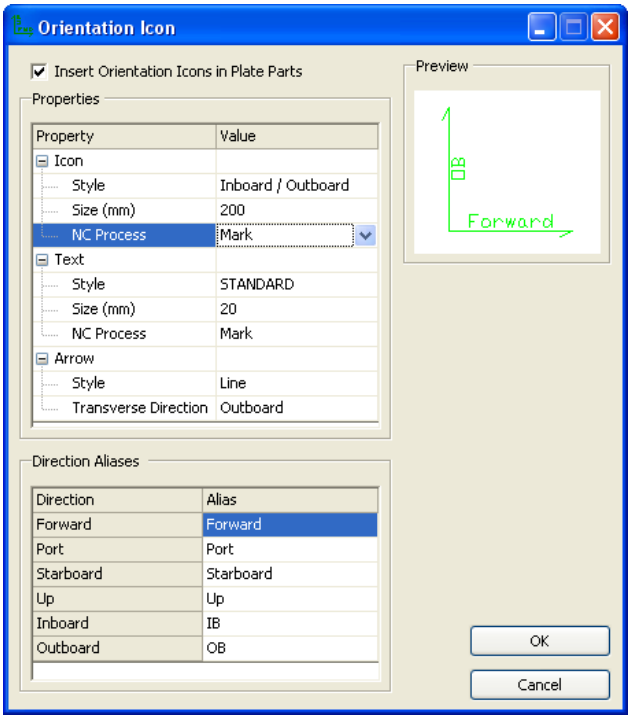
Menu ..... Structure > User-Defined Attributes
Toolbar..... None
Permissions ... Structure > Manager > Edit Structural Stocks
Procedure..... User-Defined Attributes (page 49)

Opens the User-Defined Attributes window, letting you create user-defined attributes and apply them to structural parts and stocks. User-defined attributes allow you to add custom fields to ShipConstructor.

Structure > Plates > Orientation Icon

Menu ..... Structure > Plates > Orientation Icon
Toolbar..... None
Permissions ... Structure > Manager > Edit Plate Orientation Icons
Procedure..... Orientation Icon (page 72)

Opens the Orientation Icon window, letting you create and edit the orientation icon standard. Any changes to the standard will affect all instances of the orientation icon in the model. ShipConstructor automatically places the orientation icon in the lower left corner of a part.



Insert Orientation Icons in Plate Parts – If this is checked, the orientation icon will appear on parts when you create new plate parts. You can override this in the Plate Part Properties window.

Icon

Style – The set of direction keywords to include in the orientation icon: Starboard/Port Only, Inboard/Outboard Only, or Both. The placement of the keywords is fixed.

Size – The size (length of each side) of the orientation icon.

NC Process – The NC process of the icons lines when NC cutting the part: Mark or NoProcess.

Text

Style – The text style used for the direction keywords (see Text Styles (page 35)).

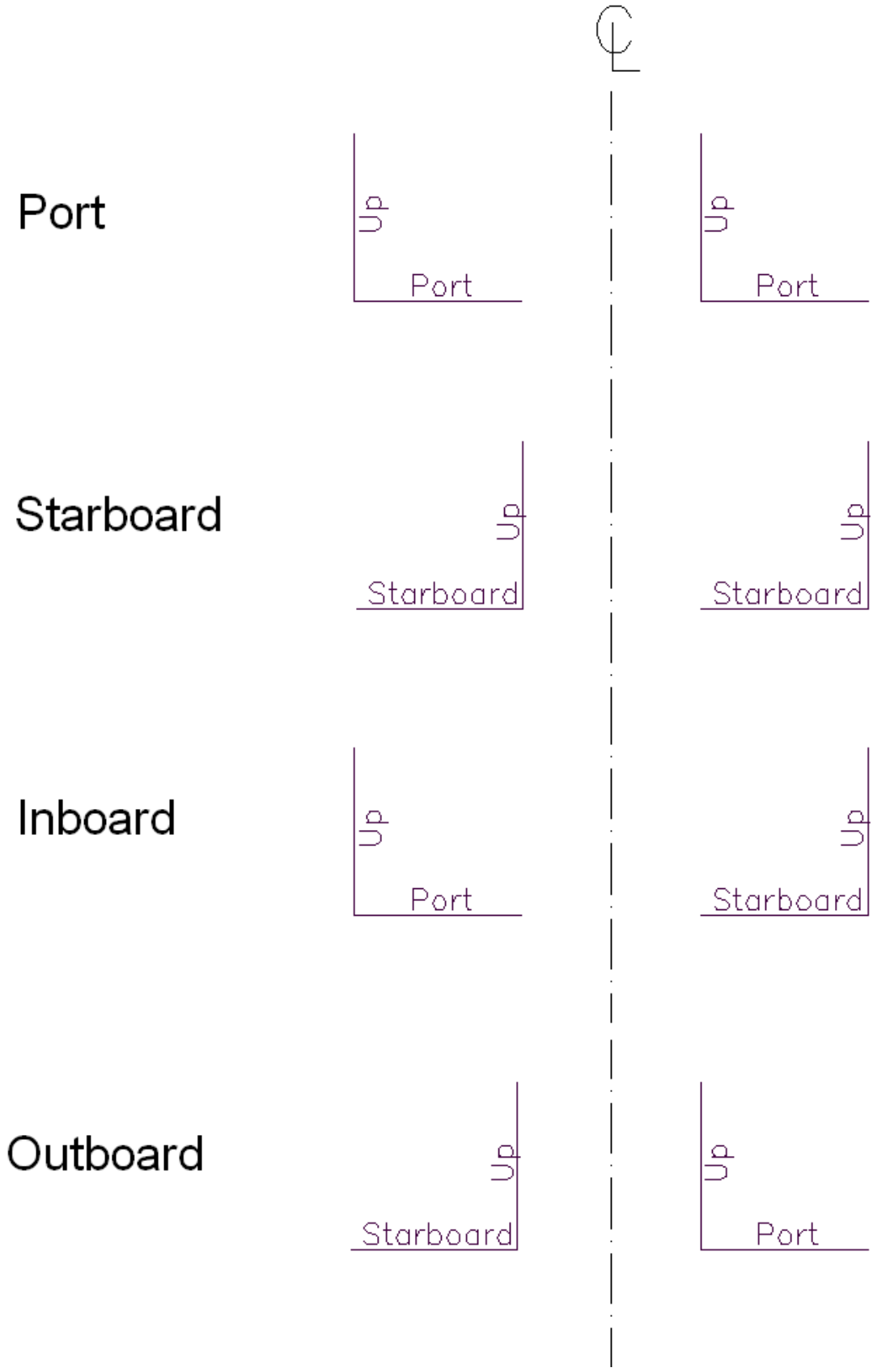
Size – The size of the text used for the direction keywords.

NC Process – The NC process of the text when NC cutting the part: Mark or NoProcess.

Arrow

Style – The style of the arrow: None, Line, or Triangle.

Transverse Direction – The direction of the orientation icon: Port, Starboard, Inboard or Outboard.


Port
Up
Port
Up
Port
Starboard
Up
Starboard
Up
Starboard
Inboard
Up
Port
Up
Starboard
Outboard
Up
Starboard
Up
Port

To specify the text used for direction keywords (in any language), set the Direction Aliases:

Forward

Port

Starboard

Up

Inboard

Outboard

Preview – A preview of how the orientation icon will look.

Structure > Plates > Markline Styles

Menu ..... Structure > Plates > Markline Styles

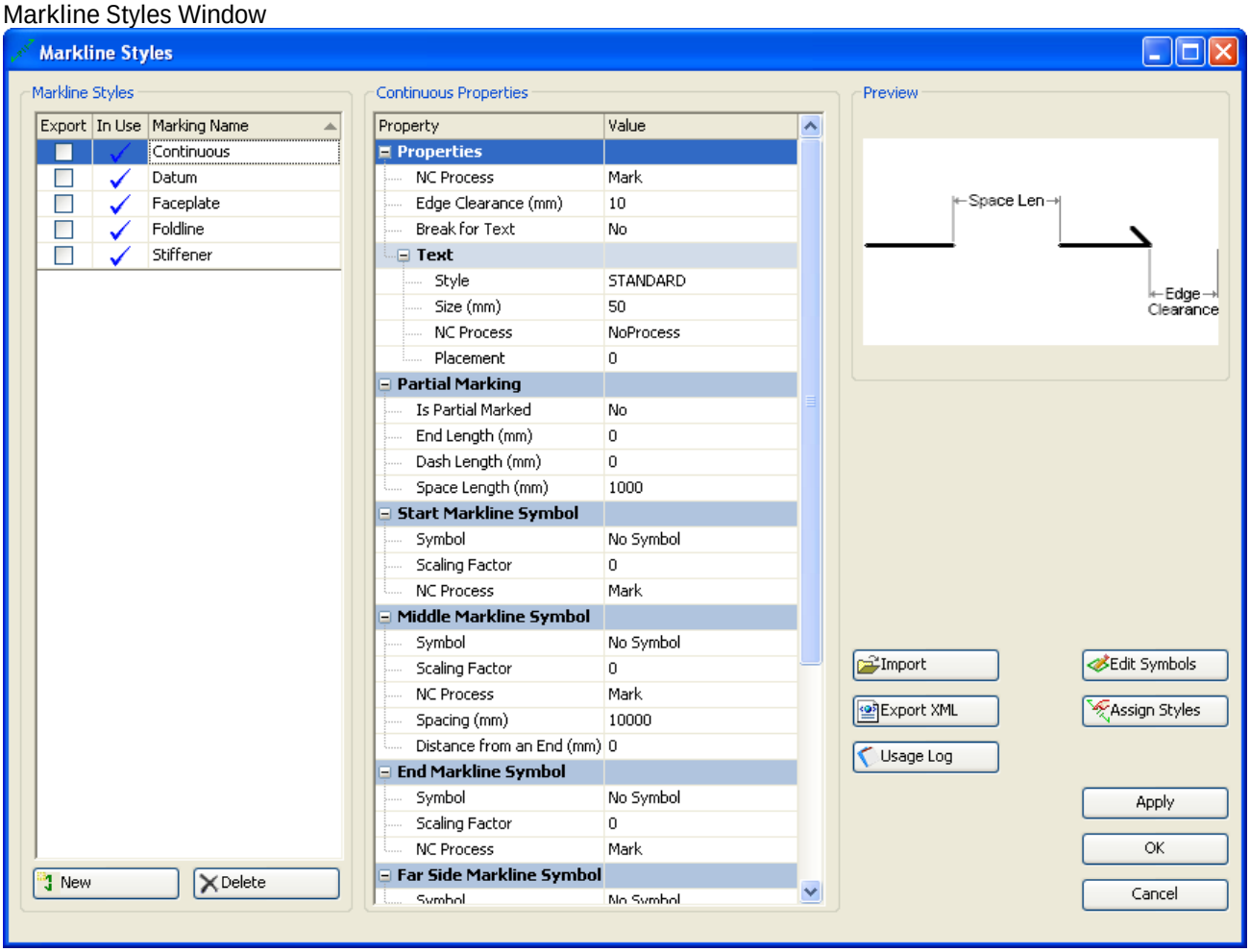
Toolbar.....

Permissions ... Structure > Manager > Edit Plate Markline Styles

Procedure..... Markline Styles (page 73)

Opens the Markline Styles window, letting you create, edit, and delete markline styles.

Markline Styles Window



Markline Styles – Lists available markline styles.

New – Creates a new markline style.

Delete – Deletes the selected markline style. You cannot delete Markline Styles that are in use.

Style Properties – Displays the properties for the selected markline style.

Properties

NC Process – The NC process of the icons lines when NC cutting the part: Mark or NoProcess.

Edge Clearance – The distance from the edge of the plate to the line. This includes holes.

Break for Text – If Yes, the line is broken apart where the line name is placed.

Text

Size – The size of the text label associated with this markline.

NC Process – The NC process of the text when NC cutting the part: Mark or NoProcess.

Placement – The distance above the line to place the text. If the Break For Text is Yes then the distance is measured from the center of the text to the line. If No, the distance is measured from the bottom of the text to the line. The markline text can be moved anywhere along the markline by dragging its grip point, but it will always be placed this distance away from the line.

Partial Marking

Is Partial Marked – If Yes, the line is broken into segments using the partial marking parameters. If No, the line will be continuous.

End Length – The length of line to leave on the end.

Dash Length – The length of the dash.

Space Length – The length of the spaces between dashes.

Start Mark Symbol

Symbol – The mark symbol to use for the start of the markline. Use None to not show a symbol.

Size – The scale of the start mark symbol.

NC Process – The NC process of the start symbol when NC cutting the part: Mark or NoProcess.

Middle Mark Symbol

Symbol – The mark symbol to use for the middle of the markline. Use None to not show a symbol.

Size – The scale of the middle mark symbol.

NC Process – The NC process of the middle symbols when NC cutting the part: Mark or NoProcess.

Spacing – The maximum distance between symbols. If this distance is 0, or greater than the length of the markline, then middle mark symbols will not be placed at regular intervals along the markline. Instead they will be placed in default locations, as determined by Distance from an End.

Distance from an End – The distance from the start and from the end to begin placing middle symbols. Middle symbols will be placed inward from these points toward the center of the line using spacing. If this is 0, middle symbols will be placed first in the center of the line and then use the spacing outward. If Spacing is 0, or greater than the markline length, then setting this to 0 will place a single middle mark symbol on the center of the markline. If Spacing is 0, or greater than the markline length, then setting this to a value greater than 0 will place one middle mark symbol on either end of the markline at the specified distance from the ends.

End Mark Symbol

Symbol – The mark symbol to use for the end of the markline. Use None to not show a symbol.

Size – The scale of the end mark symbol.

NC Process – The NC process of the end symbol when NC cutting the part: Mark or NoProcess.

Far Side Mark Symbol - This symbol is drawn when the markline is to be actually marked on the far side of the plate (the underside when the plate is lying on the NC table). Alternatively, this far side indication can be done using the Far Side Text option.

Symbol – The mark symbol to use for the far side symbol on the markline. Use None to not show a symbol.

Size – The scale of the far side mark symbol.

NC Process – The NC process of the far side mark symbol when NC cutting the part: Mark or NoProcess.

Spacing – The maximum distance between symbols.

Use Partial Marking – Yes – for production the line will be dashed using the partial marking settings. You cannot have partial marking and far side partial marking. ShipConstructor will ensure that only one is set to Yes.

Near Side Text – This label helps indicate which side of the plate to mark the line. The label appears after the text label on the line.

Indicator – Text to place on the markline to indicate that the markline is marked on the correct side of the plate. If you do not want to use this as an indicator leave the field blank.

Size – Height of the near side text. The style of the text is the same as the Text > Style.

NC Process – The NC process of the near side text when NC cutting the part: Mark or NoProcess.

Far Side Text - This label helps indicate which side of the plate to mark the line. The label appears after the text label on the line.

Indicator – Text to place on the markline to indicate that the markline is marked on the other side of the plate. If you do not want to use this as an indicator leave the field blank.

Size – Height of the far side text. The style of the text is the same as the Text > Style.

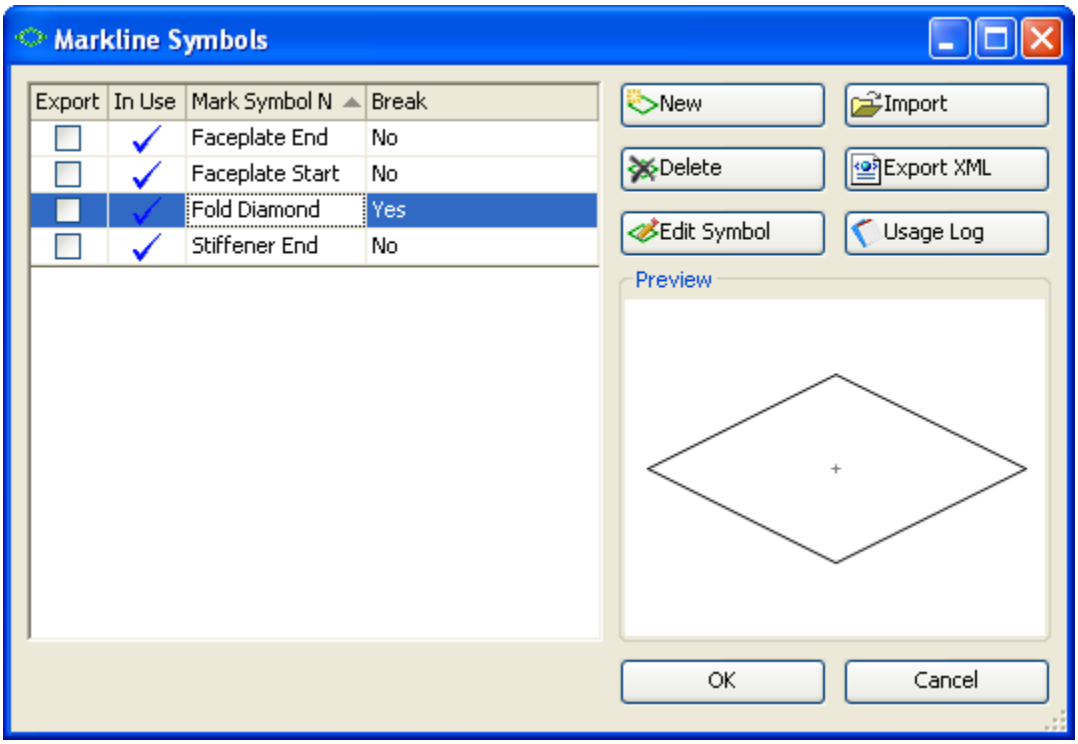
NC Process – The NC process of the far side text when NC cutting the part: Mark or NoProcess.

Edit Symbols – Opens the Mark Symbols Window (page 350), letting you create and edit markline symbols.

Assign Styles – Opens the Assign Markline Styles Window (page 351), letting you control which marklines styles can be used for stiffeners, flange foldlines, corrugated plate foldlines, and datum lines.

Mark Symbols Window

Click Edit Symbols in the Markline Styles window to open this window. Mark symbols are completely customizable. They are lines, arcs, polylines, circles placed along a markline.

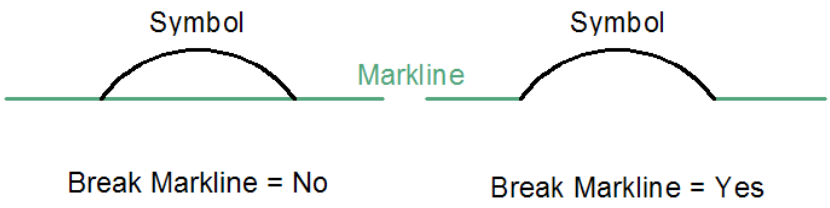


Mark Symbols

In Use – A check indicates that a markline style is using this symbol.

Symbol Name – The name of the markline symbol. Names must be unique.

Break Markline – Yes indicates that the markline is broken where the symbol is located.

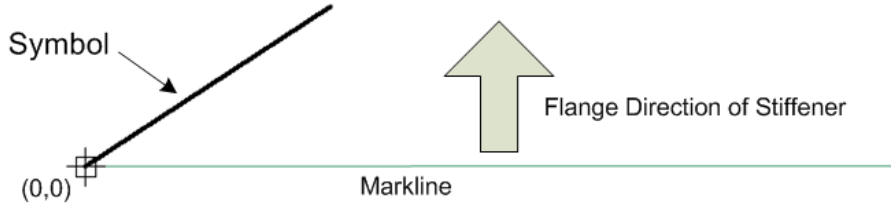


New – Creates a new mark symbol.

Delete – Deletes the selected markline symbol. You cannot delete symbols that are in use.

Edit Symbol – Opens the selected symbol in an AutoCAD drawing enabling you to edit the symbol.

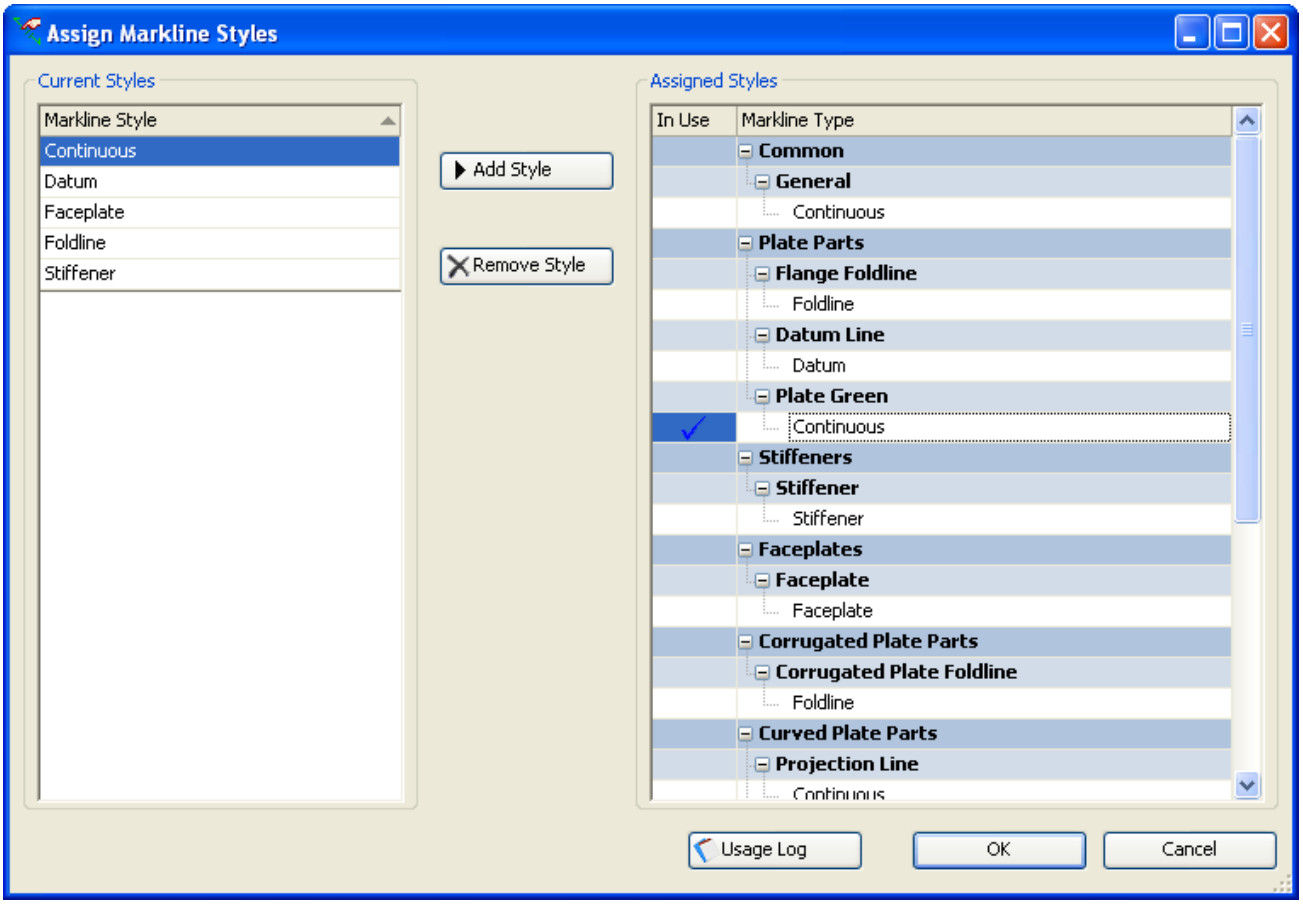
When editing the symbol, it is important to know the orientation of the markline.



When the symbol is used at the end of the markline then the symbol is mirrored.

Assign Markline Styles Window

Click Assign Styles in the Markline Styles window to open this window. Marklines can be used in many places. The Assign Markline Styles window lets you select markline styles when creating each of the different markline types. You must assign markline styles to the markline types to be able to use markline styles.



Current Styles – Lists all the markline styles.

Assigned Styles – Lists the types of marklines and the styles associated with those types.

In Use – Indicates with a checkmark that a markline style is used in a part.

Markline Type – The fixed list of markline types ShipConstructor supports. Under each markline type is the markline style that can be selected on each instance of the markline.

Add Style – Adds the selected Current Styles to the selected Markline Types.

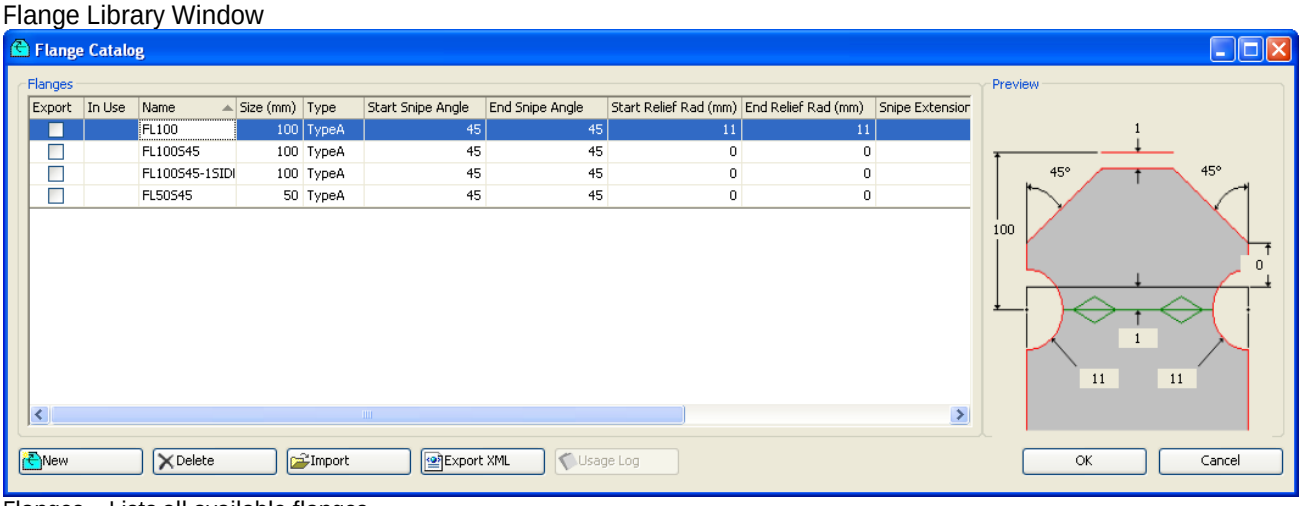
Remove Style – Removes the selected styles from the markline types. You cannot remove a style that is in use.

Structure > Plates > Flange Catalog

Menu ..... Structure > Plates > Flange Catalog
Toolbar..... None
Permissions ... Structure > Manager > Edit Plate Flange Standards
Procedure..... Flange Catalog (page 79)

Opens the Flange Library window, letting you create, edit, import, and export plate flanges.

Flange Library Window



Flanges – Lists all available flanges.

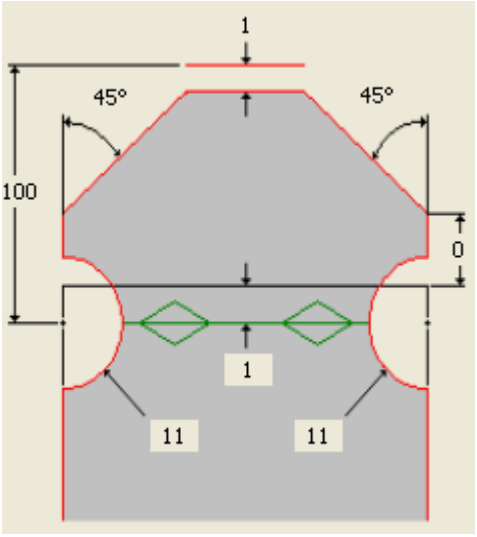
Export – Checked flanges will be exported when you click Export XML.

In Use – Indicates with a checkmark which flanges are currently in use in the project.

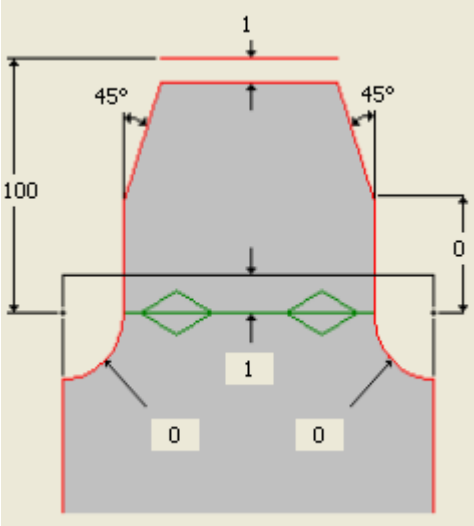
Name – The name of the flange.

Size – The distance from the bend to the end of the flange.

Type – The type of flange.



Type A Flange



Type B Flange

Start / End – When you add a flange to a plate part and the flange definition has different settings for the start and end, ShipConstructor asks you to select the start by picking a point on the part.

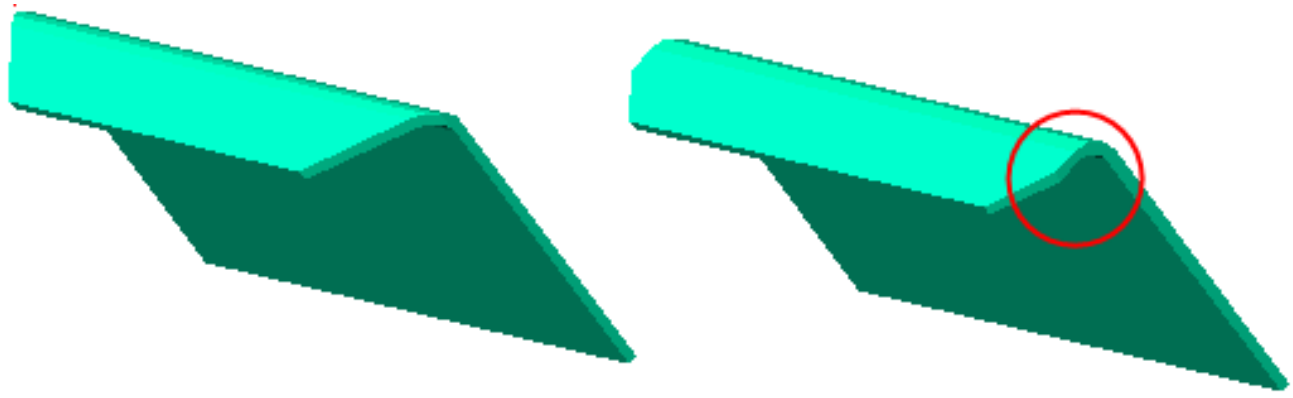
Start Snipe Angle – the angle to snipe the flange at the start of the flange.

End Snipe Angle – the angle to snipe the flange at the end of the flange.

Start Relief Rad – the radius of the relief arc inserted into the expanded production geometry at the start.

End Relief Rad – the radius of the relief arc inserted into the expanded production geometry at the end.

Snipe Extension – The amount of stock to add to the flange between the R-end (where the radius of curvature ends) and the beginning of the snipe. This can provide extra grip for the flange bending machine, especially when the flange has snipe ends. If Snipe Extension is 0, no stock is added to the flange.



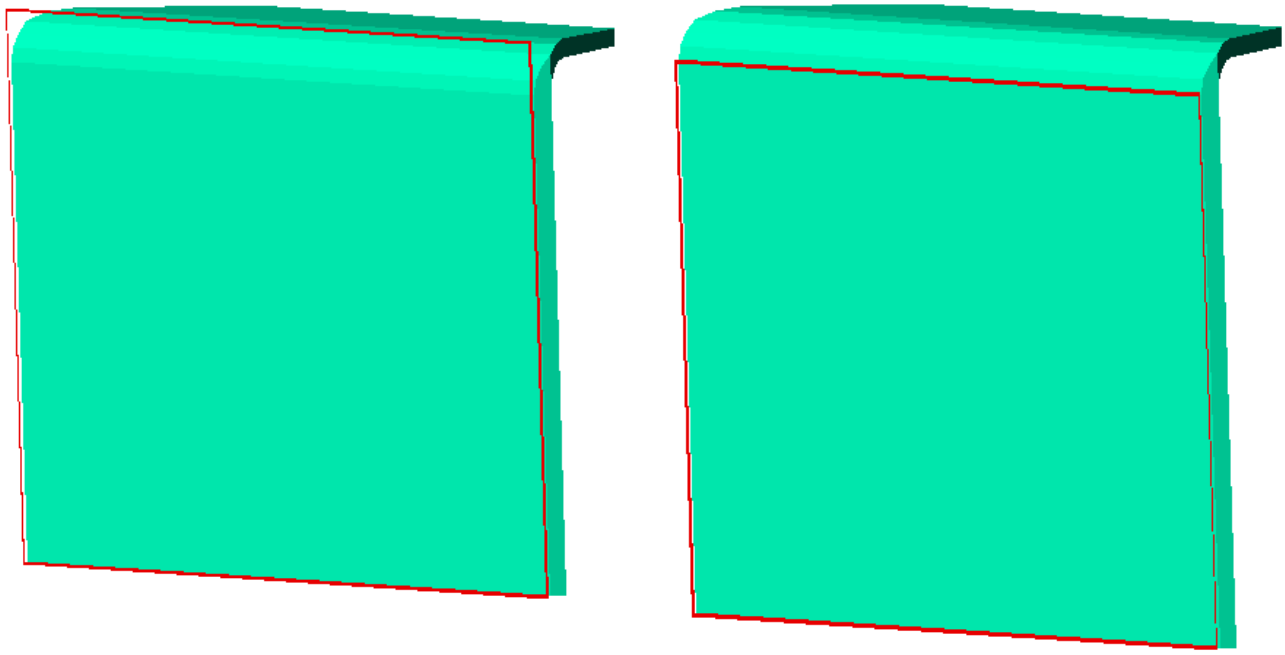
A flange with zero (left) and non-zero (right) Snipe Extension

Flange Correction – The amount to move the edge of the expanded flange. This is a multiple of the plate thickness.

Foldline Correction – The amount to move the edge of the plate to represent the expanded flange. This is a multiple of the plate thickness.

Use Rend – Creates the flange so its radius of curvature begins at the end of the original plate. This is useful when a flange is attached to a stiffener and you want the web of the flange to be the same height as the flange (for example, so the flange can be welded along the entire length of the stiffener). If Use Rend is off, the flange is created so the top of the flange is aligned with the end of the original plate.

Note: When Use Rend is off, the uncorrected foldline will appear along the edge of the original plate. When Use Rend is on, the uncorrected foldline will be offset by twice the flange radius from the original plate edge.



A flange with Use Rend off (left) and on (right)

Preview – Displays an image of the flange Type with all values indicated. When you select or edit a field in the table, the corresponding value becomes highlighted in the Preview.

New – Creates a new flange.

Delete – Deletes the selected flange.

Import – Lets you import flanges from another project or from an XML file that was exported from another project.

Export XML – Exports the checked Export flanges to an XML file.

Structure > Plates > Corner Treatments

Menu ..... Structure > Plates > Corner Treatments
Toolbar..... None
Permissions ... Structure > Manager > Edit Plate Corner Treatments
Procedure..... Corner Treatments (page 81)

Opens the Corner Treatments window, letting you create and edit corner treatments.

Structure > Plates > Green Standards

Menu ..... Structure > Plates > Green Standards
Toolbar..... None
Permissions ... Structure > Manager > Edit Plate Green Standards
Procedure..... Plate Green Standards (page 83)

Opens the Plate Green Standards window, letting you create and edit plate green standards. Plate green is the addition of extra material to a plate to handle fit up problems. Plate green also allows you to specify a markline to indicate the original edge.

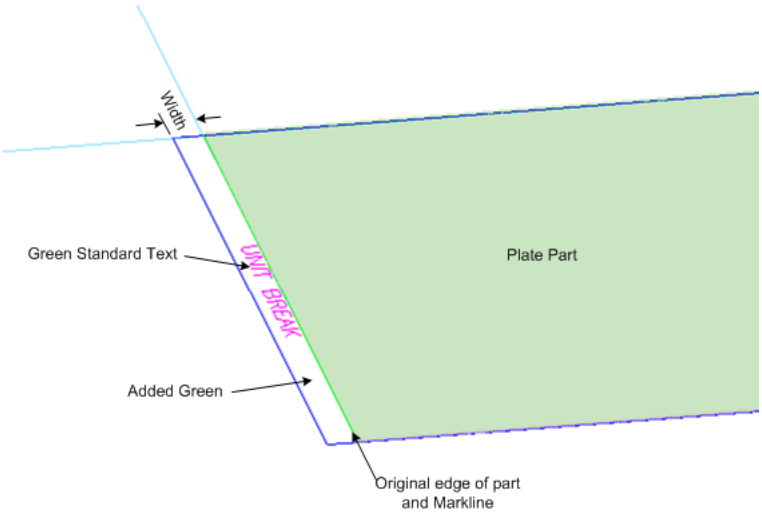
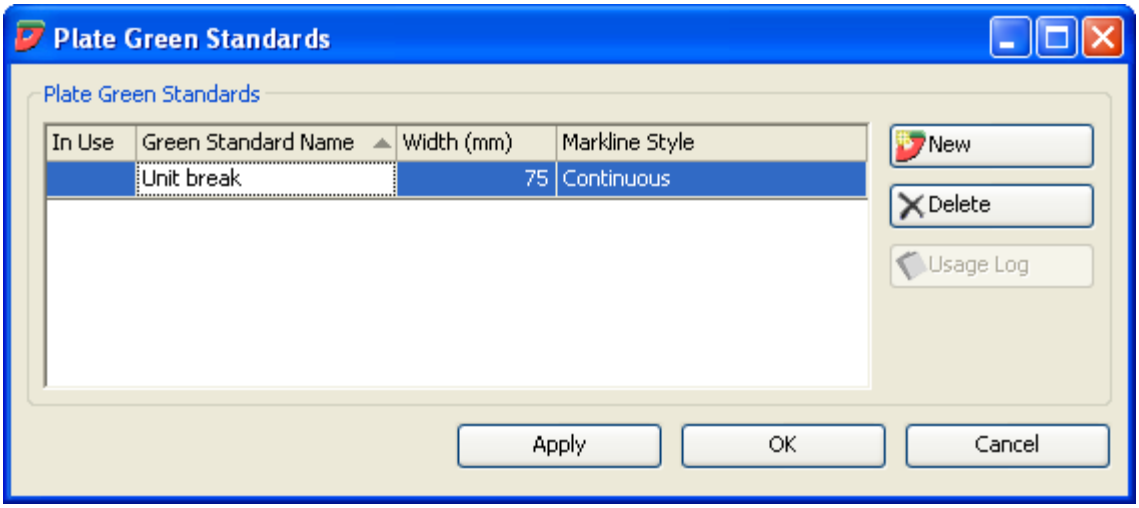


Plate Green added for a unit break

Plate Green Standards Window



Plat Green Standards -- The list of available plate green standards that can be applied to plate parts.

In Use – The check indicates that the green standard is used in a plate part. In Use standards cannot be deleted.

Green Standard Name – The name of the green standard. This text will be placed on the part on the green portion.

Width – The width of the green portion.

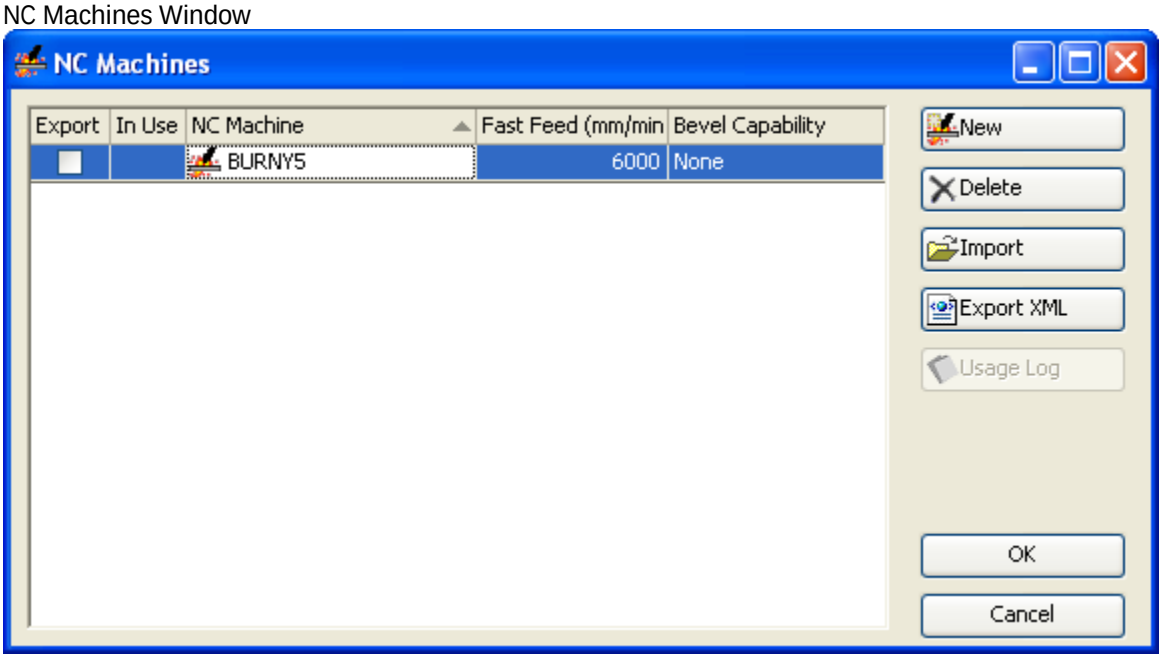
Markline Style – The style of markline to mark on the original edge of the plate. The available markline styles is determined in the Plate > Plate Green - Markline Type in the Assign Markline Styles Window (page 351).

Structure > Plates > NC Machines

Menu ..... Structure > Plates > NC Machines
Toolbar..... None
Permissions ... Structure > Manager > Edit Plate NC Machines
Procedure..... NC Machines (page 90)

Opens the NC Machines window, letting you create and edit NC machines.

NC Machines Window



Export – Checked NC machines will be exported when you click Export XML.

NC Machine – The name of each NC machine.

Fast Feed – The NC machine's fast travel feed rate.

Bevel Capability – The NC Machine's capability to cut bevel angles. None, ThreeTorch (or single pass), or Multipass. Multipass requires multiple passes to cut a K type bevel.

In Use – Indicates with a checkmark which NC machines are currently assigned to nests.

New – Creates a new NC machine.

Delete – Deletes the selected NC machine.

Import – Lets you import NC machines from another project or from an XML file that was exported from another project.

Export XML – Exports the checked Export NC machines to an XML file. That can be imported into another project.

Structure > Plates > Bevel Standards

Menu ..... Structure > Plates > Bevel Standards
Toolbar..... None
Permissions ... Structure > Manager > Edit Plate Bevel Standards
Procedure..... Bevel Standards (page 85)

Opens the Bevel Standards window, letting you create and edit bevel standards.

Structure > Plates > Bevel Settings

Menu ..... Structure > Plates > Bevel Settings
Toolbar..... None
Permissions ... Structure > Manager > Edit Plate Bevel Standards
Procedure..... Automatic Bevels (page 88)

Opens the Bevel Settings window, letting you configure automatic beveling.

Structure > Profiles > Endcut Catalog

Menu ..... Structure > Profiles > Endcut Catalog
Toolbar..... None
Permissions ... Structure > Manager > Edit Profile Endcut Definitions
Procedure..... Endcuts (page 92)

Opens the Endcut Definitions window, letting you create and edit profile stock endcuts.

Endcuts are made from multiple cuts. A cut is made from a standard shape that is extruded and then subtracted from the end of the stiffener. A cut can be directed through the web or through the flange. Endcuts cannot be made from drawings anymore, so you will need to determine the type of cuts needed to reproduce the from-drawing cut.

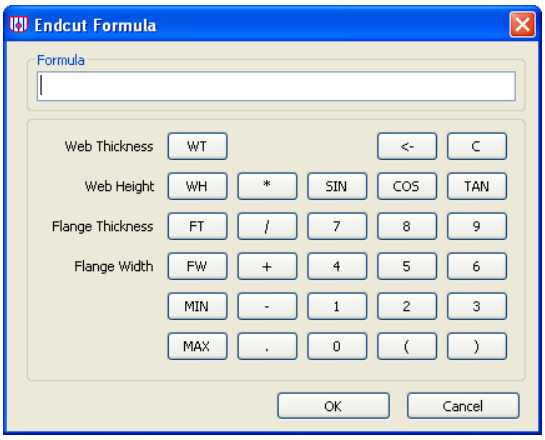
The Endcut Catalog now includes a dynamic preview that shows the current endcut on one of the stiffener stocks that it has been assigned to. If the preview shows nothing, right-click in the preview window and click Zoom Extents.

Cut values can be decimal values or a formula. Clicking shows the Formula window.

Endcut Formulas

Endcut formulas allow you to create a single endcut for multiple size stocks. You can simply type the endcut formula in the value field or use the calculator. The calculator shows you all the possible operations and keywords.

Note: Currently, negative values must be entered inside parenthesis as (-37.5)



Cut Properties

Applied To

Web – If this is checked, the cut is extruded across the web of the stiffener.

Flange – If this is checked, the cut is extruded across the flange of the stiffener.

Size -- Varies depending on the cut.

Origin

Location X – Distance from the end of the stiffener to place the origin of the cut. In most cases this will be 0.

Location Y – Distance from the base (moldline point) to place the origin of the cut.

Trim Adjustment – Use this option to adjust the cut's position in case the stiffener has trim(s).

None – No adjustment (default)

Full Dimension – Adjusts the cut's position so that one of the specified dimensions (Length, Width, Nose Width, Radius) is fully realized by extending the cutout shape. The outcome varies depending on the cut.

First Contact – Adjusts the cut's position to the location where one of its key points first intersects the solid. The key points vary depending on the cut.

Angled Dimension – Applies to snipe and double snipes. Measures Width and Length along the angle of trim.

Depth -- The cut depth allows you to apply a cut to only a portion of the stiffener like only one flange of an I-beam.

Cut Depth Start – The start length of the cut. If the cut is a web cut, then the Depth Start is across the web with 0 being the moldline position.

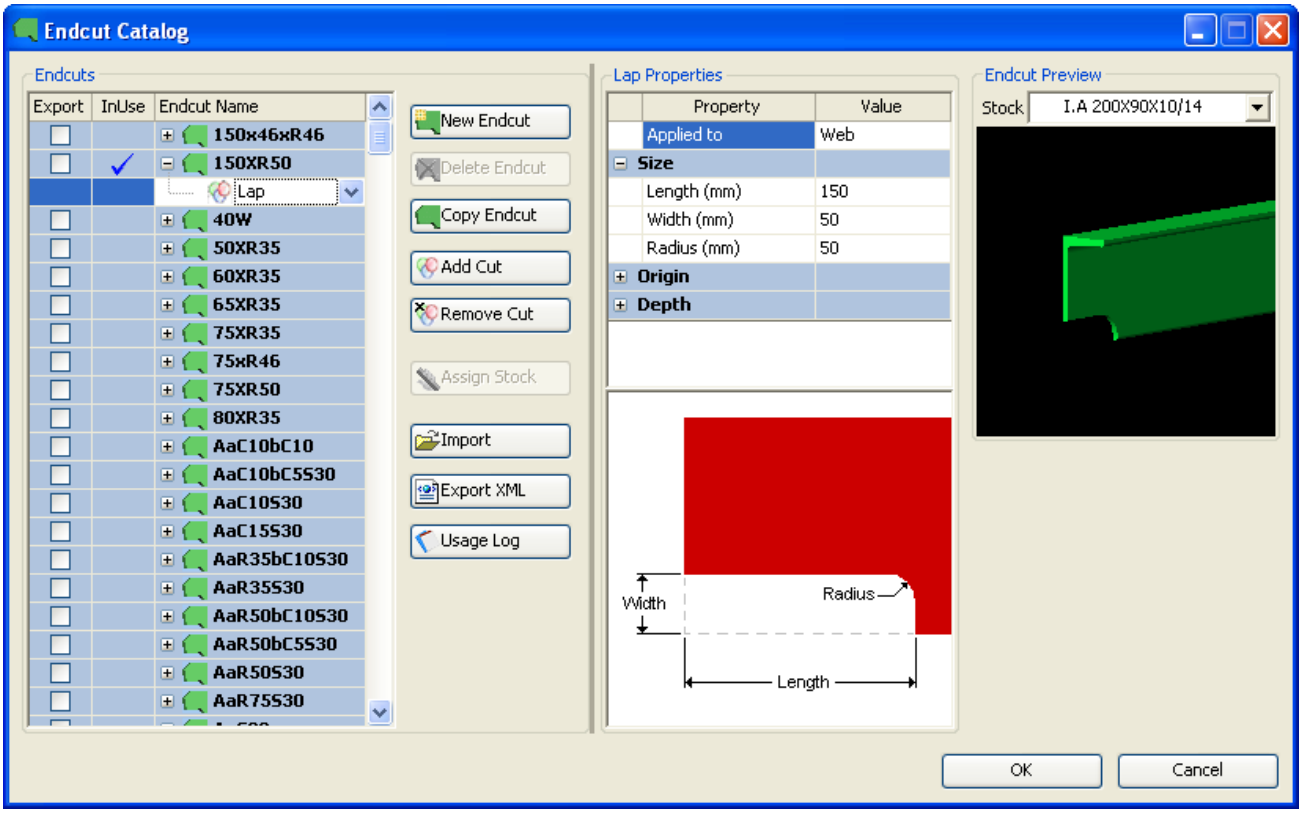
MIN – Leftmost extent of the cross section. (For Applied to: Flange it is the bottommost extent.)

MAX – Rightmost extent of the cross section. (For Applied to: Flange it is the topmost extent.)

Cut Depth End – The end length of the cut.

To open the endcut catalog window

1. In Manager, choose Structure > Profiles > Endcut Catalog (page 358), or click on the button on the Manager toolbar.



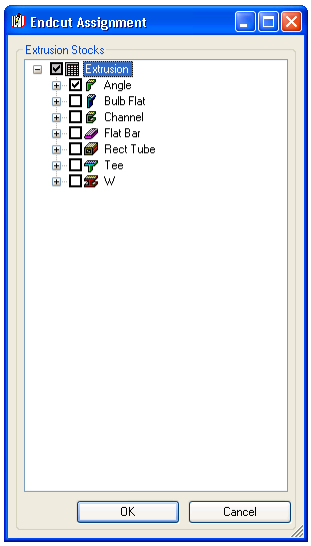
To add a cut to an endcut

1. Select an endcut from the Endcuts list.
2. Click the Add Cut button.

To assign a stock to an endcut

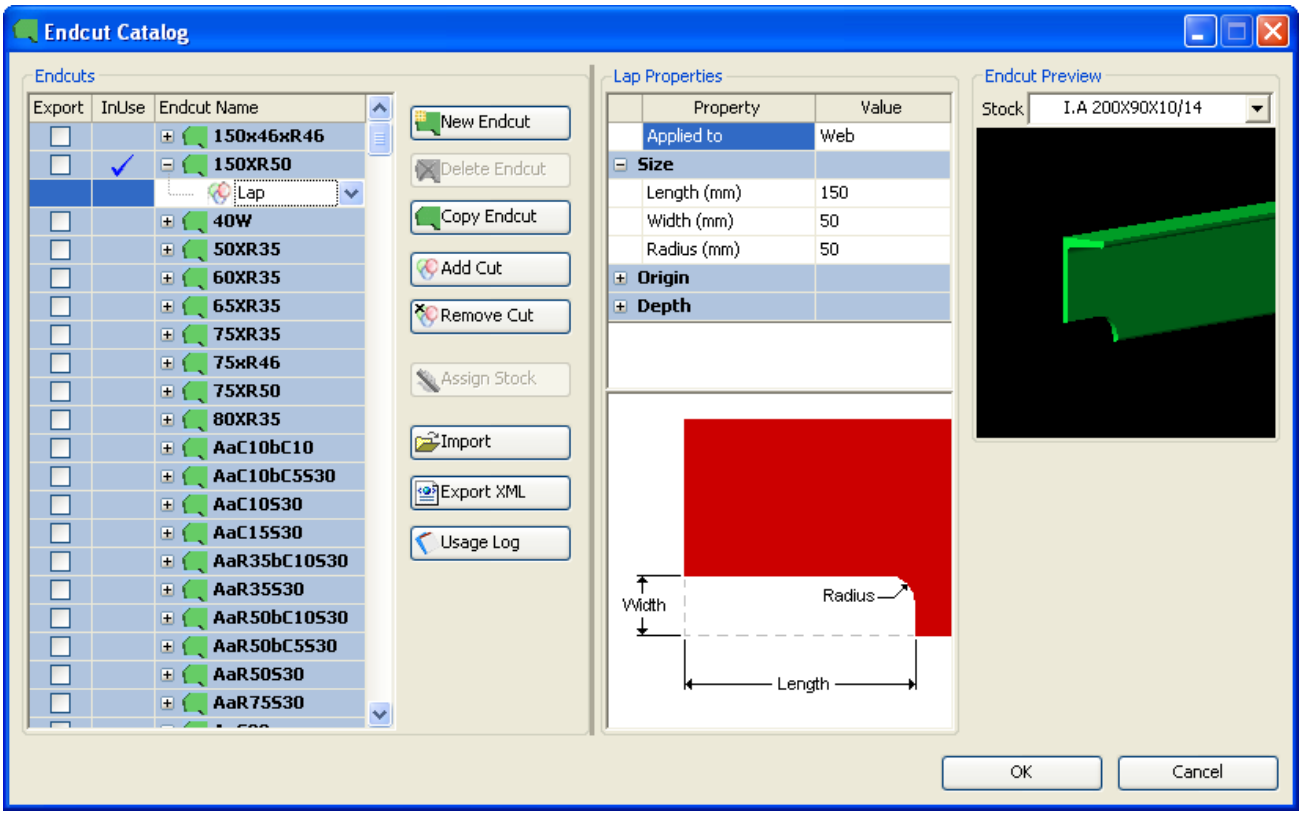
Endcuts need to be assigned to the specific profile stocks that they can be applied to. This controls the structural modeler from making mistakes and selecting a wrong endcut for a stiffener.

1. Select Endcuts from the Endcuts list.
2. Click Assign Stock. Existing assignments will be checked already.



3. Check the stocks you want to allow the selected endcuts to be applied to.

Endcut Definitions Window



Endcuts – Lists available endcuts and lets you create and edit endcuts.

In Use – Indicates with a checkmark which endcuts are currently in use in the project.

Export – Checked endcuts will be exported when you click Export XML.

Endcut Name – The name of the endcut. Cuts are indicated as sub elements of the Endcut Name.

New Endcut – Creates a new endcut.

Delete Endcut – Deletes the selected endcut. You cannot delete an endcut that is in use by a part.

Add Cut – Adds a cut to the selected endcut.

Remove Cut – Removes the selected cut from its endcut.

Assign Stock – Opens the Endcut Assignment window for the selected endcut, letting you select the profile stocks to which the endcut can be applied.

Import – Lets you import flanges from another project or from an XML file that was exported from another project.

Export XML – Exports the checked Export flanges to an XML file.

Cut Properties – Lists and lets you edit the properties of the selected cut.

Applied to – Defines whether the cut is applied to the Web or Flange of the profile stock part.

Size, Origin, Depth – Lets you specify the size, origin, and depth of the cut.

The Size values are different for each cut type.

When you select or edit a value, the image below the Properties indicates its effect. The image is different when selecting or editing Size values, Origin values, or Depth values.

To specify a constant value, enter the value. To specify a variable value (based on the web or flange dimensions of the profile stock), click to open the Endcut Formula window and enter a formula.

Endcut Preview – A preview image of the selected endcut applied to the selected Stock. Only stocks assigned to the end cut are available to preview for the selected endcut.

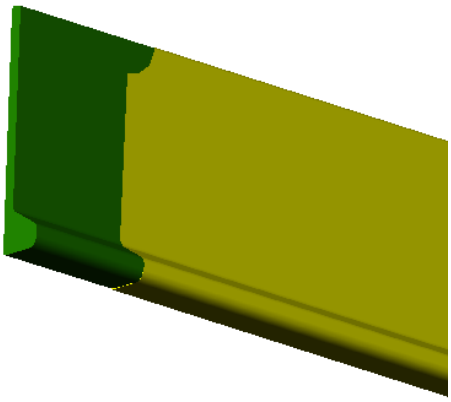
Right Click Menu – There is a right click menu for the Preview. This menu lets you change the view of the preview.

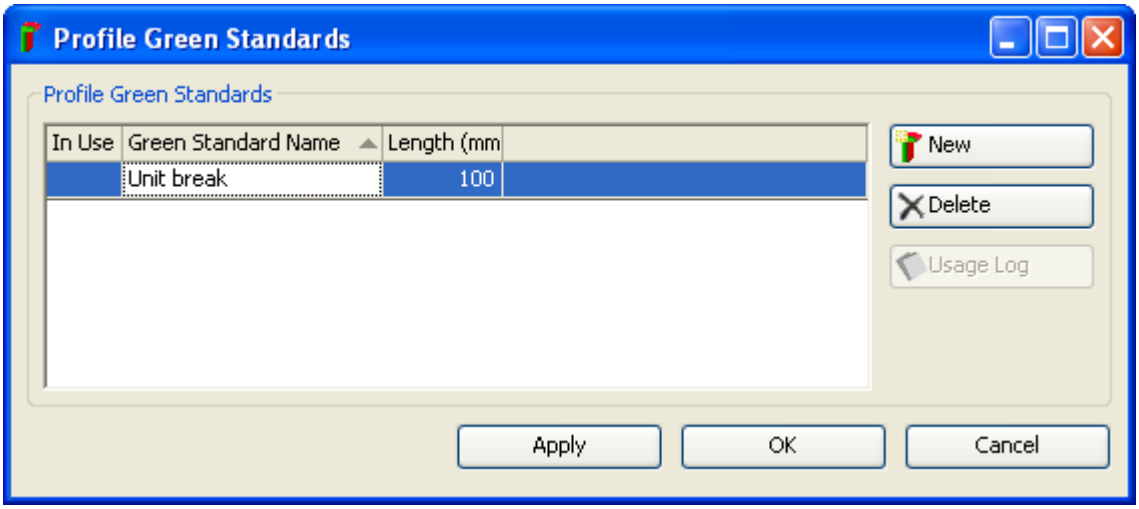
Stock – The profile stock used in the Endcut Preview.

Structure > Profiles > Green Standards

Menu	Structure > Profiles > Green Standards
Toolbar	None
Permissions	Structure > Manager > Edit Profile Green Standards
Procedure	Profile Green Standards (page 95)

Green is defined as added stock on a part that allows for some correction during fit up. Typically this happens at erection joints or unit breaks. Adding green keeps any endcut or trims and extends the length of the stiffener from its longest point. Changing a green standard will affect all parts using that standard.





Profile Green Standards

In Use – Indicates with a checkmark if the green standard is used in parts.

Green Standard Name – The name of the green standard. This name is not displayed in production drawings but can be seen in reports. Names must be unique.

Length – The amount of green to add to stiffeners that have this green applied.

New – Creates a new green standard.

Delete – Deletes the selected green standard. You cannot delete green standards that are used in parts.

Structure > Standard Parts

Menu ..... Structure > Standard Parts
Toolbar..... None
Permissions ... Structure > Manager > Edit Structural Standards
Procedure..... Standard Parts Definition (page 97)

Opens the Standard Parts window, letting you create and edit standard (structural) parts.

Structure > Dynamic Marking Blocks

Menu ..... Structure > Dynamic Marking Blocks
Toolbar..... None
Permissions ... Structure > Manager > Edit Structural Standards
Procedure..... Dynamic Marking Block Standards (page 101)

Opens the Dynamic Marking Block Catalog to create and edit dynamic marking blocks.

Piping > Stock Catalog

Opens the Pipe Stock Catalog, letting you create and edit pipe library items. See the Pipe manual for details.

Piping > Pipe Benders

Opens the Pipe Bending Machines window, letting you create and edit pipe bending machines. See Pipe Bending Machines in the Pipe manual for details.

Piping > User-Defined Attributes

Opens the User-Defined Attributes window, letting you create and edit user-defined attributes for pipe parts and stocks. See User-Defined Attributes (page 49).

HVAC > Stock Catalog
Opens the HVAC Catalog Stock Editor, letting you create and edit end treatments, HVAC stocks, catalogs, specs, and connections. See the HVAC manual for details.

HVAC > Sheet Stocks
Opens the Sheet Stocks window, letting you create and edit sheet stocks. See Sheet Stocks in the HVAC manual for details.

HVAC > User-Defined Attributes
Opens the User-Defined Attributes window, letting you create and edit user-defined attributes for HVAC parts and stocks. See User-Defined Attributes (page 49).

Equipment > Equipment Class Editor
Opens the Equipment Class Editor, letting you create and edit equipment items. See the Pipe manual for details.

Penetrations > Standards
Opens the Penetration Standard Editor, letting you create and edit penetration standards. See the Pipe manual for details.

Database Utilities > Raw Data
Opens the Database Raw Data window, letting you display and edit the raw data contained in the project database. See the Project Management manual for details.

Window >
Lets you switch to open windows in Manager.

> Note: If no windows are open, the Window menu is empty.

Toolbars

Equipment Toolbar
Insert Equipment Part

See the Pipe manual for details.

Hull Toolbar
See the Hull manual for details.

Hull Curve Toolbar
See the Hull manual for details.

Hull Expand Toolbar
See the Hull manual for details.

Hull Pin Jig Toolbar
See the Hull manual for details.

Hull Porcupine Toolbar
See the Hull manual for details.

Hull Stringer Toolbar

See the Hull manual for details.

Hull Surface Toolbar

See the Hull manual for details.

Labels Toolbar

Label Properties

Penetrations Toolbar

Create Penetration

See the Equipment and Penetrations manual for details.

ShipConstructor Toolbar

Navigator

See ShipConstructor > Navigator (page 368)

Model Link

See ShipConstructor > Model Link (page 383)

Manager

See ShipConstructor > Manager (page 383)

Report

See the Project Management manual for details.

Structural Tools Toolbar

Mirror about Centerline

See SC Utilities > Mirror about Centerline (page 461)

3D to 2D

See SC Utilities > 3D to 2D (page 457)

Structure Toolbar

New Planar Group

See Structure Page (page 371)

Move Planar Group

See SC Structure > Move Planar Group (page 449)

Mark Group Intersections

See SC Structure > Mark Group Intersections (page 393)

Swap Construction Line

See SC Structure > Construction Line > Swap Construction Line (page 441)

Replace Construction Line Geometry

See SC Structure > Construction Line > Replace Construction Line Geometry (page 442)

Structure Plate Flyout

Create Plate

See SC Structure > Plate > New

Edit Plate

See SC Structure > Plate > Edit

Create Corrugated Plate

See SC Structure > Corrugated Plate > New

Edit Corrugated Plate

See SC Structure > Corrugated Plate > Edit

Structure Flange Plate Flyout

Flange Plate

See SC Structure > Plate > Add Flange

Edit Flange

See SC Structure > Plate > Edit Flange

Structure Green Flyout

Add Green

See SC Structure > Plate > Add Green

Open Green Manager

See SC Structure > Plate > Edit Green

Structure Part Objects Flyout

Add Objects to Part

See SC Structure > Plate > Add Objects (page 404)

Remove Objects from Part

See SC Structure > Plate > Remove Objects (page 406).

Manage Cutouts

See SC Structure > Plate > Manage Cutouts (page 406)

Edit Component

See SC Structure > Parts > Edit Component (page 435)

Add Manual Cutout

See SC Structure > Parts > Add Manual Cutouts (page 438)

Structure Corner Treatment Flyout

Insert Corner Treatment

See SC Structure > Plate > Corner Treatment > Insert

Remove Corner Treatment

See SC Structure > Plate > Corner Treatment > Remove

Display Corner Treatments

See SC Structure > Plate > Corner Treatment > Display (page 408)

Manage Datum Lines

See SC Structure > Plate > Manage Datum Lines (page 410)

Manage Weld Shrinkage Icon

See SC Structure > Plate > Manage Weld Shrinkage Icon (page 410)

Structure Stiffener Flyout

Create Stiffener

See SC Structure > Stiffener > New (page 412)

Create Stiffener from Points

See SC Structure > Stiffener > New from Points

Edit Stiffener

See SC Structure > Stiffener > Edit

Insert Cutout

See SC Structure > Stiffener > Cutout > Insert

Edit Cutouts

See SC Structure > Stiffener > Cutout > Edit (page 417)

Trim Stiffener

See SC Structure > Stiffener > Trim

Structure Faceplate Flyout

Create Faceplate

See SC Structure > Faceplate > New

Edit Faceplate

See SC Structure > Faceplate > Edit

Part List

See SC Structure > Parts > List (page 431)

Insert Standard Part

See SC Structure > Insert Standard Part (page 444)

Structure Display Toolbar

Structure Display

See SC Structure > Show / Hide Options

Toggle Show Production

See SC Structure > Show / Hide Options

Toggle Show Solid

See SC Structure > Show / Hide Options

Toggle Show Draft

See SC Structure > Show / Hide Options

Utilities Toolbar

3D View

See SC Utilities > 3D Viewpoint

Activate UCS

See SC Utilities > Activate UCS

Hide Objects

See SC Utilities > Hide Objects (page 454)

Unhide Objects

See SC Utilities > Unhide Objects

Snap

See SC Utilities > Snap

Layers Flyout

Move Geometry to Layer

See SC Utilities > Layer > Move Geometry to

Copy Objects to Layer

See SC Utilities > Layer > Copy Geometry to

Activate Layer

See SC Utilities > Layer > Activate

Deactivate Layer

See SC Utilities > Layer > Deactivate (page 459)

Tool Path

See SC Utilities > Toolpath (page 460)

Dihedral Angle

See SC Utilities > Dihedral Angle (page 461)

Reload Drawing

See SC Utilities > Reload Drawing (page 462)

ShipConstructor Menu

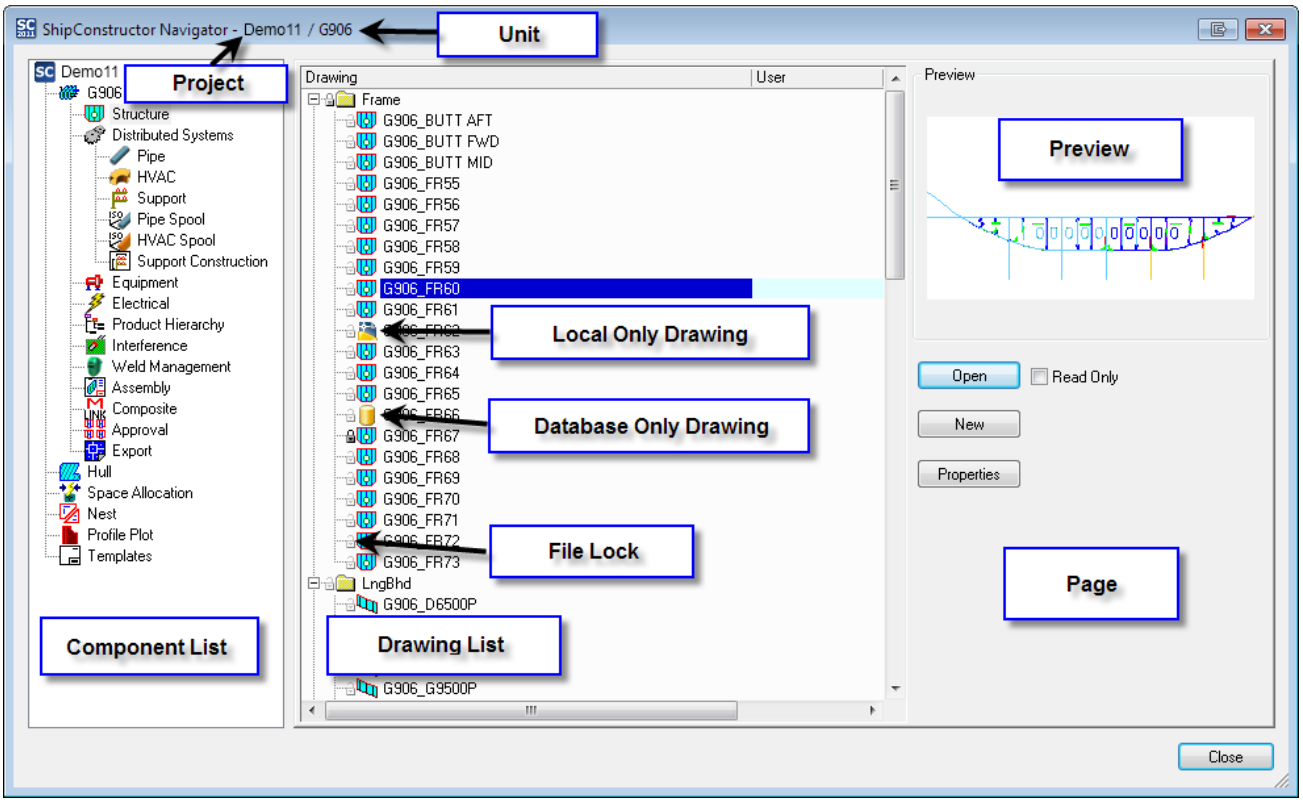
ShipConstructor > Navigator

Button .....
Ribbon ..... ShipConstructor tab > Navigation panel > Navigator
Menu ..... ShipConstructor > Navigator
Toolbar.....
Command ..... SCNAVIGATE
Permissions ... None

Opens the Navigator, letting you create and open drawings and perform other drawing-related functions.

Note: You must use Navigator to create, open, or move drawings so that ShipConstructor can keep track of them. Do not use Windows Explorer, or ShipConstructor will not be able to properly associate the drawings with the project.

Navigator



Component List – Each project is divided into several components or folders. The components of the current project are listed in the Component List. The first component is the project name. Within the project are one or more units and several project-wide components (Hull, Space Allocation, Nest, Profile Plot, Templates). Within each unit are several unit-related components (Structure, Pipe, HVAC, Supportm, Spools, Support Construction, Equipment, Product Hierarchy, Interference, Weld Management, Assembly, Composite, Approval, Export). When you select an item in the Component List, the associated drawings are listed in the Drawing List and the associated buttons are displayed on the Page.

To customize the Component List, right-click within it and choose Customize to open the Customize the Navigator window. Select the items you want to appear in the Component List and click OK.

Drawing List – The Drawing List lists the drawings associated with the selected item in the Component List.

Each drawing in the Drawing List is displayed with the following:

- A lock icon, (locked) or (unlocked) – Clicking the lock icon locks (or unlocks) the drawing so it cannot (or can) be modified. (If clicking the lock icon does not seem to work, you may not have permissions to lock and unlock the appropriate drawing files.)
- Drawing name – Right-click on the drawing name to display a menu of commands.
- User – The name of the user who currently has the drawing open (as read/write).
- Last Date Modified – The date that the drawing was last modified. (You may need to scroll across or resize Navigator to see this information.)
- Preview – The Preview is a preview image of the drawing selected in the Drawing List.
- Local Only Drawing icon – The drawing exists on the local file system and not on the server.
- Database Only Drawing icon – The drawing exists on the server and not on the local file system.

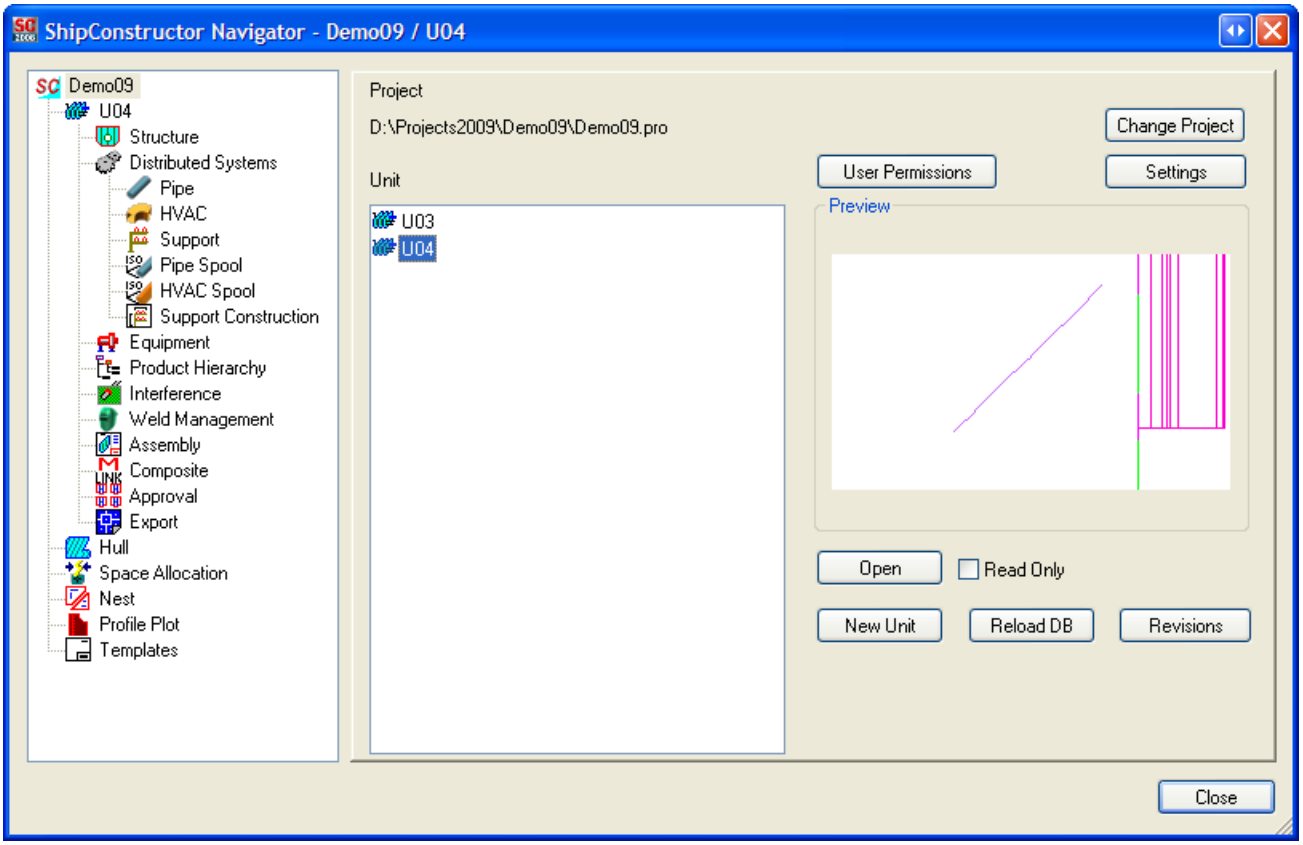
Sometimes a preview image does not appear. If ShipConstructor has automatically modified a drawing (for example, if you make a change in one drawing, ShipConstructor may automatically update other drawings), a preview image does not appear until the next time you open and save the drawing.

Page – The Page buttons let you perform functions related to the selected item in the Drawing List. The following items are found on all pages:

- Read Only – If this is turned on when you open a drawing, you cannot modify and save the drawing. If Read Only is turned off when you open a drawing, you can modify and save the drawing.
- Close Current – If this is turned on when you open a drawing, ShipConstructor will automatically close the current drawing. This option is only available when you are working in multiple document interface (MDI) mode. ShipConstructor has been optimized for single drawing compatibility mode (SDI), so we recommend using SDI mode. To switch between MDI mode and SDI mode, type “SDI” at the AutoCAD command line. Use “1” for SDI.

Other items are different for each page (see below).

Project Page



The Project page lets you create, open, and modify unit drawings and perform project-related functions.

Change Project – Opens the Register Project window, letting you switch to another project.

User Permissions – Opens the User Permissions window (See General > User Permissions (page 332)).

Settings – Opens the Project Settings window (See General > Project Settings (page 327)).

Open – Opens the selected unit drawing.

New Unit – Adds a unit to the current project. See Create Units (page 107) for details.

Reload DB – Reloads the settings from the project database into ShipConstructor. Use Reload DB whenever you make changes to the project settings in Manager.

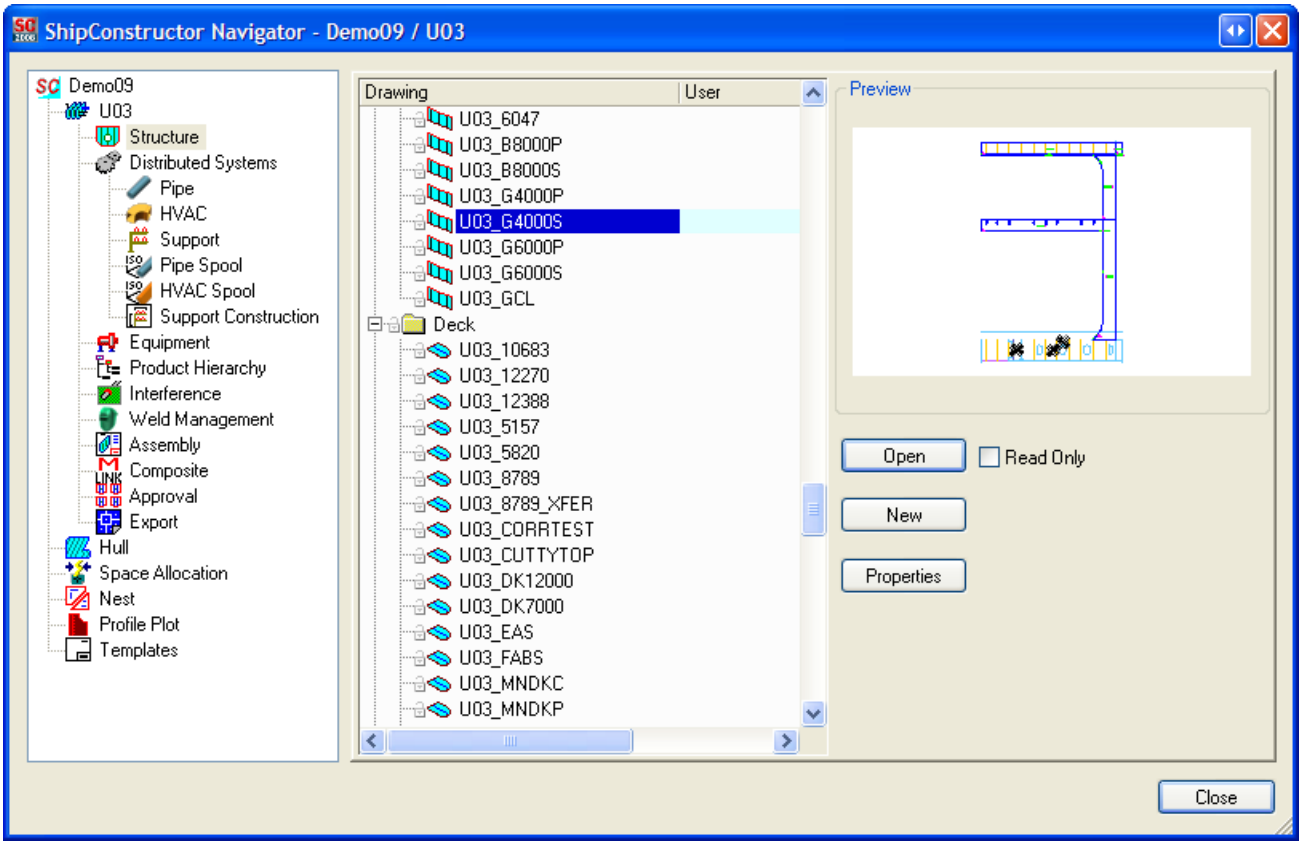
Revisions – Displays changes made to the current project. See ShipConstructor > Revisions for details.

Unit Page

The Unit page lets you open a unit drawing.

Structure Page

The Structure page lets you create, open, and modify structural drawings (planar group drawings and curved plate drawings) belonging to the registered unit. See Structure Modeling Concepts (page 126).



Open

Opens the selected structural drawing. If you turn on Read Only, you will not be able to modify and save changes to the drawing.

New

See ShipConstructor > Planar Group > New.

Properties

Permissions ... Structure > Create/Delete Drawings
Procedure..... Edit Planar Group Properties (page 133)

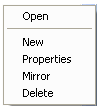
Opens the Edit Group Properties window, letting you edit the properties of the selected planar group drawing.

Delete

Permissions ... Structure > Create/Delete Drawings
Procedure..... Delete a Planar Group (page 133)

Deletes the selected planar group drawing and all the parts in it.

Right-Click Menu – Lets you open, delete, or modify the properties of the selected structural drawing or create a new structural drawing.



Pipe Page
The Pipe page lets you create, open, and modify pipe drawings. See the Pipe manual for details.

HVAC Page
The HVAC page lets you create, open, and modify HVAC drawings. See the HVAC manual for details.

Support Page
The Support page lets you create, open, and modify Support and Hanger drawings. See the Pipe manual for details.

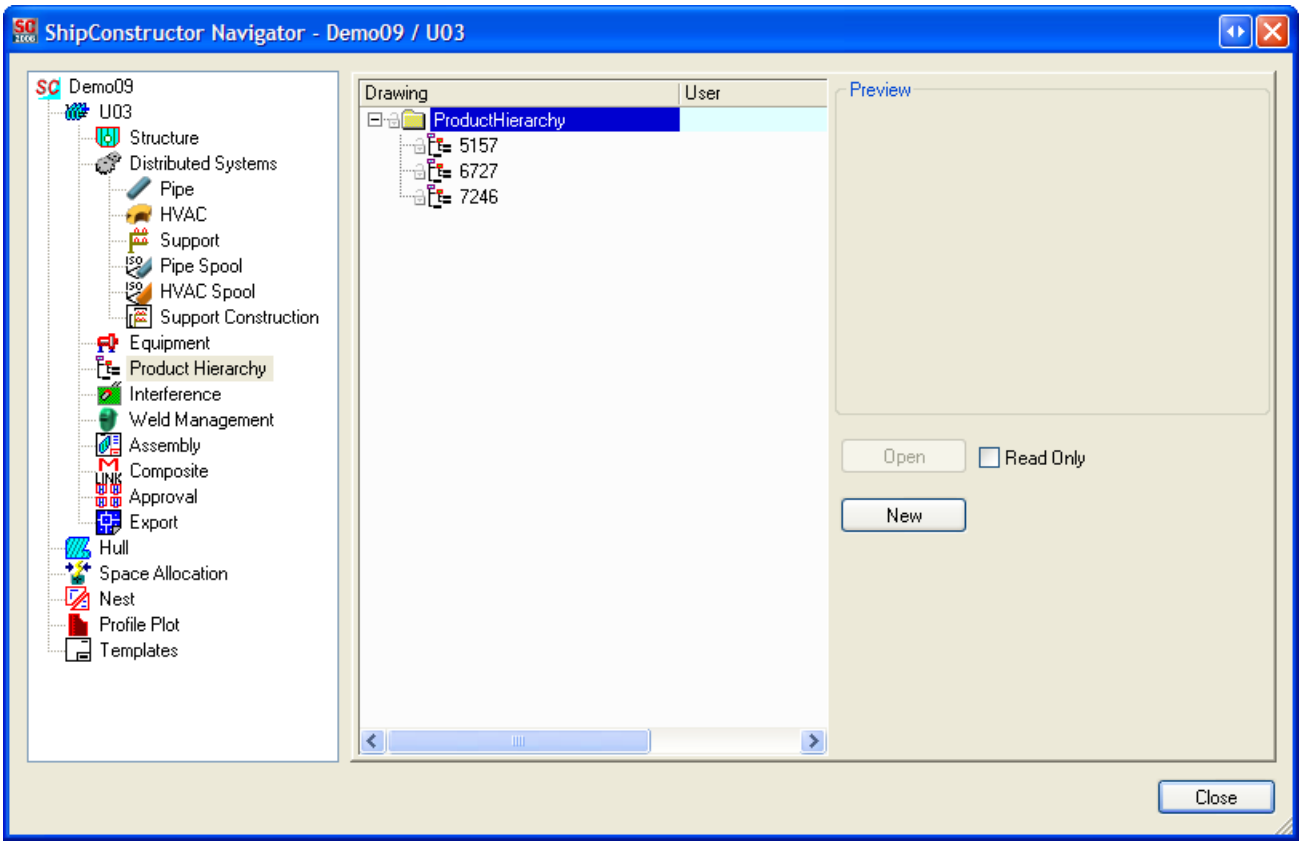
Pipe Spool Page
The Pipe Spool page lets you create, open, and modify pipe spool drawings. See the Pipe manual for details.

HVAC Spool Page
The HVAC Spool page lets you create, open, and modify HVAC spool drawings. See the HVAC manual for details.

Support Construction Page
The Support Construction page lets you create, open, and modify support construction drawings. See the Pipe manual for details.

Equipment Page
The Equipment page lets you create, open, and modify equipment drawings. See the Equipment manual for details.

Product Hierarchy Page
The ProductHierarchy page lets you create, open, rename, or delete product hierarchy drawings. See Product Hierarchies (page 108).



Right-Click Menu – Lets you open, rename, or delete the selected product hierarchy drawing or create a new product hierarchy drawing.

Open

Permissions ... None

Opens the selected product hierarchy drawing.

New

Permissions ... General > Edit Product Hierarchy
Procedure..... Create a Product Hierarchy Drawing (page 113)

Creates a new product hierarchy drawing. A product hierarchy drawing is used to visualize the parts in the unit when developing the product hierarchy tree of parts. This drawing will contain read-only copies of the parts that can be selected individually.

Rename

Permissions ... General > Edit Product Hierarchy
Procedure..... Rename a Product Hierarchy Drawing (page 115)

Lets you rename the selected product hierarchy drawing.

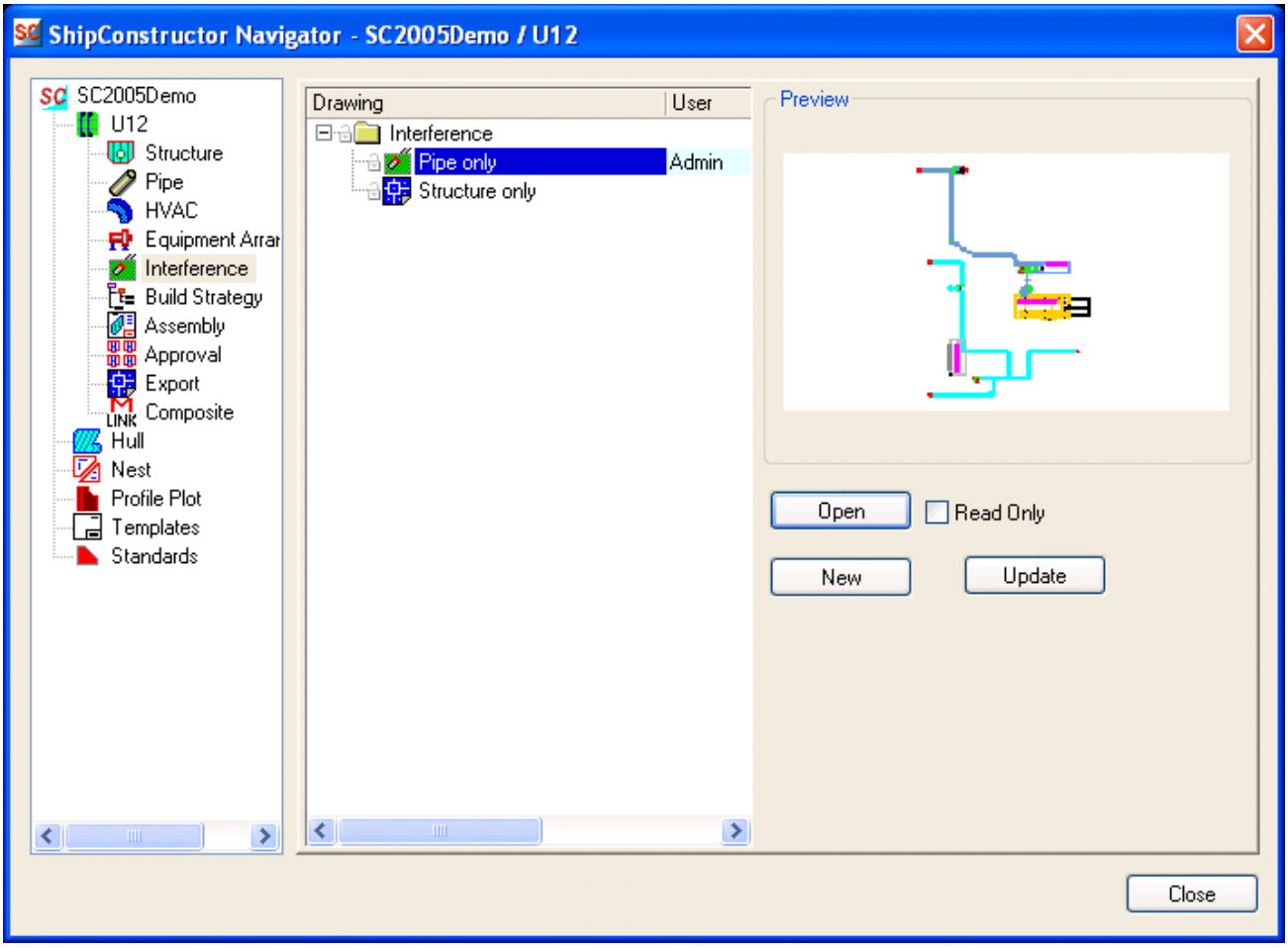
Delete

Permissions ... General > Edit Product Hierarchy
Procedure..... Delete a Product Hierarchy Drawing (page 115)

Deletes the selected product hierarchy drawing.

Interference Page

The Interference page lets you create, open, rename, or delete interference drawings.



Open

Permissions ... General > None

Opens the selected interference drawing.

New

Permissions ... General > Edit Interference

Creates a new interference drawing.

Update

Permissions ... General > Edit Interference

Updates the selected interference drawing based on the latest drawings. This only updates solids from drawings that you originally checked when you created the interference drawing.

Right-Click Menu – Lets you open, rename, update, or delete the selected interference drawing or create a new interference drawing.

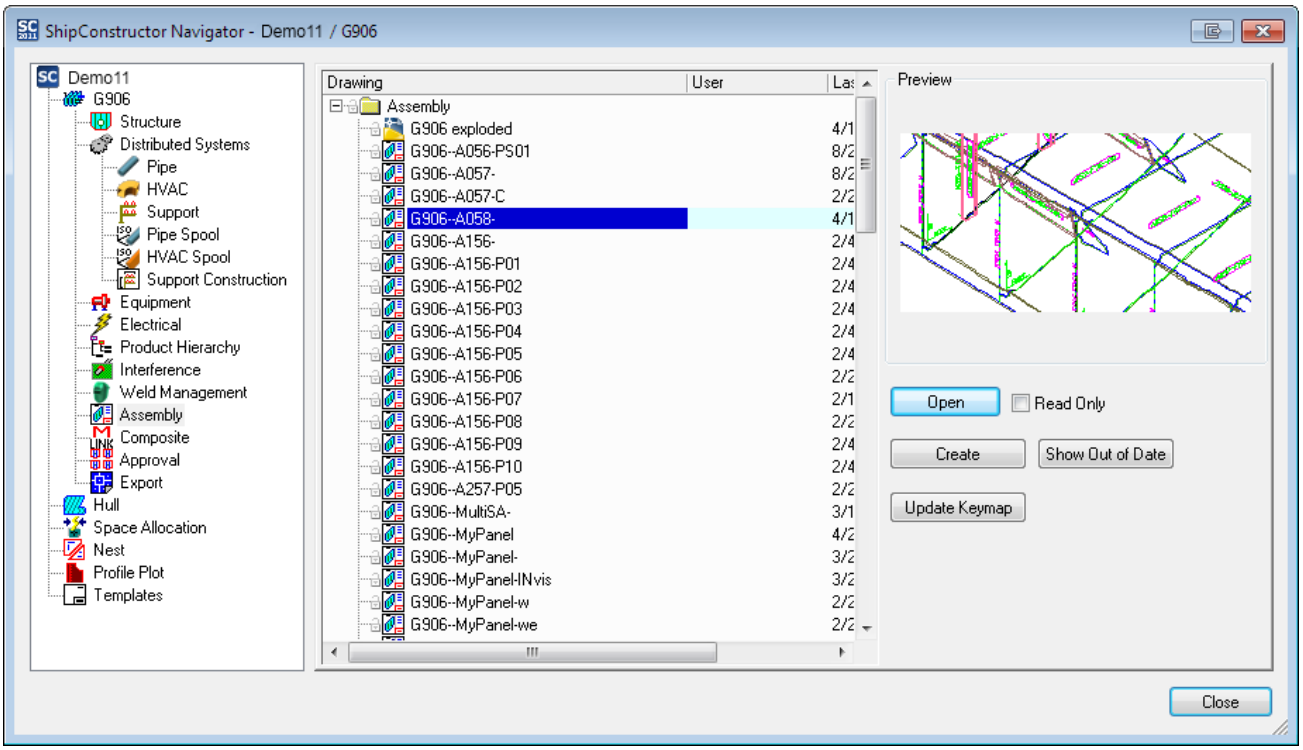
New, Rename, Update, and Delete require General > Edit Interference permissions.

Weld Management Page

The Weld Management page lets you create, open, and modify weld management drawings. See the Weld Management manual for details.

Assembly Page

The Assembly page lets you open or create assembly drawings. See Assembly Drawings (page 218).



Open – Opens the selected assembly drawing.

Create – Creates a new assembly drawing.

Update Keymap – Creates or updates the drawing KEYMAP that contains the HullTrace construction lines for the current unit. The Keymap drawing is referenced in assembly drawings as an overlay of the current assembly. It gives reference to where the assembly is located in the unit or vessel. The keymap drawing can contain any objects. It is recommended that you prepare the keymap drawing after it has been created.

Show Out of Date – Shows a ⚠ next to each drawing that has parts in it that have changed, parts that have been added or removed.

Right-Click Menu – Lets you open, rename, or delete the selected product hierarchy drawing or create a new product hierarchy drawing.

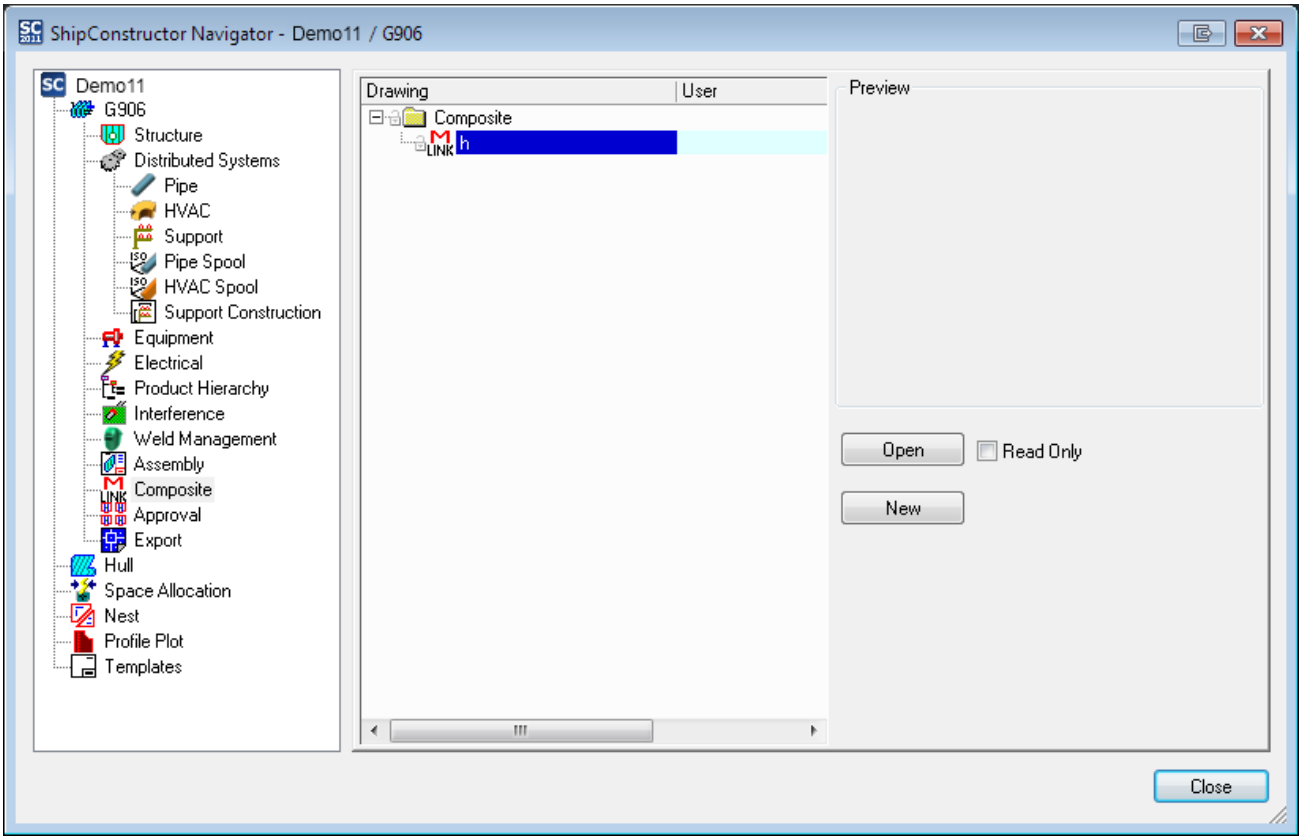
Reattach Drawing

Permissions ... Structure > Assembly Drawings – Edit

Reattaches an existing drawing to a registered assembly drawing that has its drawing file missing. The selected drawing is checked to ensure it is an assembly drawing. The assembly drawing needs to show as an assembly drawing icon in the Navigator to be elligable for reattaching.

Composite Page

The Composite Page lets you create and open composite drawings (drawings that you can use to MLink in other drawings in order to see multiple modules together). See Set Up Composite Drawings (page 16).



Open

Permissions ... None

Opens the selected composite drawing.

New

Permissions ... General > Edit Composite Drawings

Creates a new composite drawing.

Right-Click Menu
Lets you open, rename, or delete the selected composite drawing or create a new composite drawing.

Rename

Permissions ... General > Edit Composite Drawings

Lets you rename the selected composite drawing.

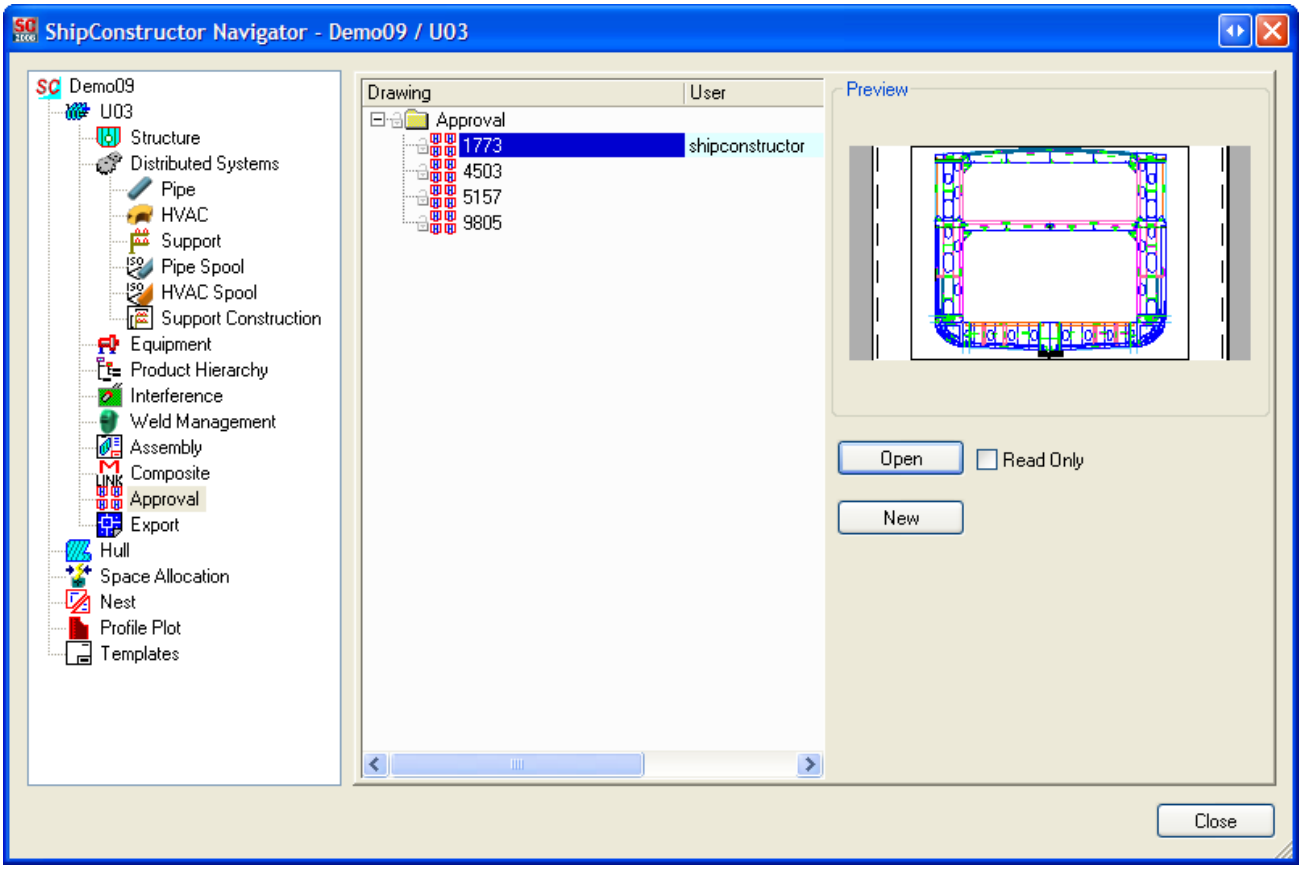
Delete

Permissions ... General > Edit Composite Drawings

Deletes the selected composite drawing.

Approval Page

The Approval page lets you create and open approval drawings.



Open

Permissions ... None

Opens the selected approval drawing.

New

Permissions ... General > Edit Approval Drawings

Creates a new approval drawing.

Right-Click Menu – Lets you open, rename, or delete the selected approval drawing or create a new approval drawing.

Rename

Permissions ... General > Edit Approval Drawings

Lets you rename the selected approval drawing.

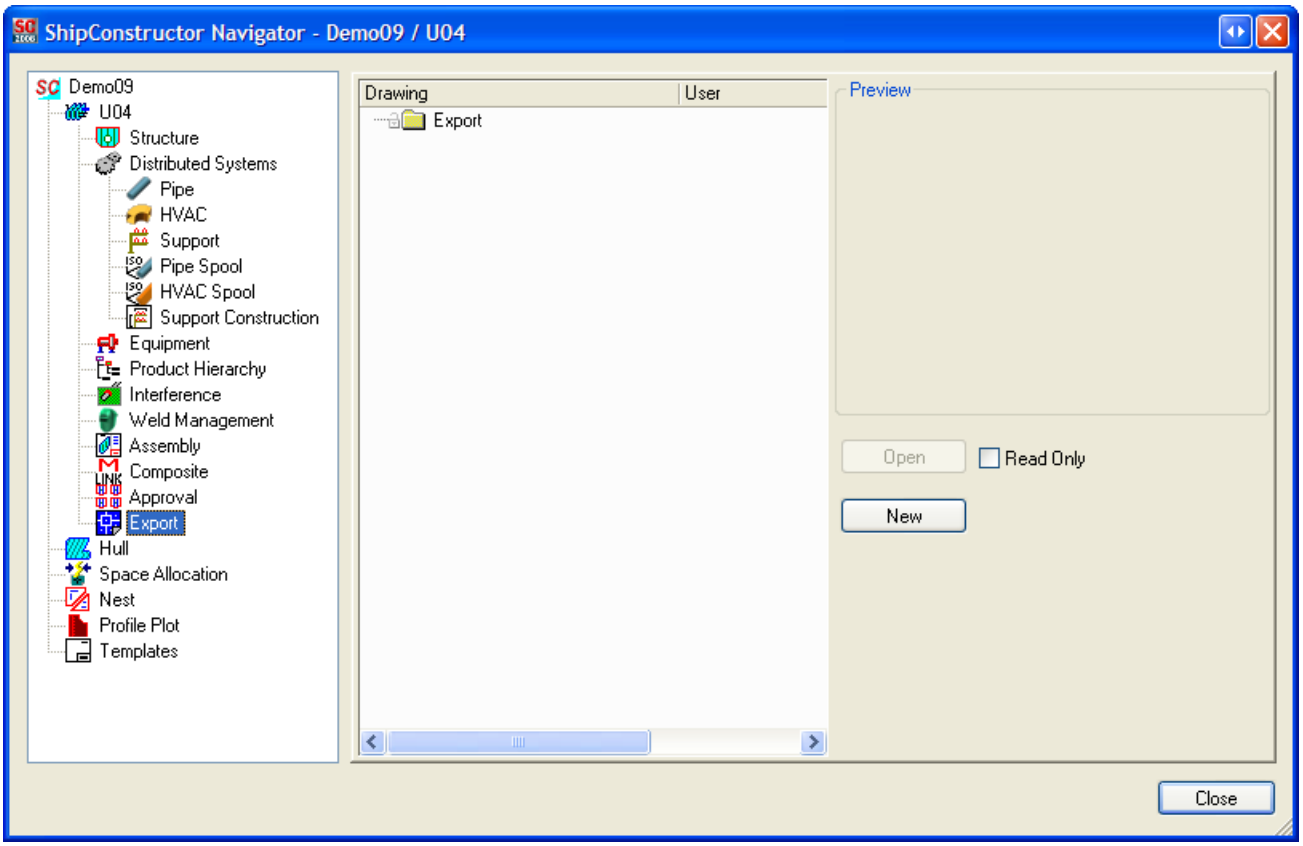
Delete

Permissions ... General > Edit Approval Drawings

Deletes the selected approval drawing.

Export Page

The Export page lets you export any number of ShipConstructor drawings to a standard AutoCAD drawing.



Open

Permissions ... None

Opens the selected export drawing.

New

Permissions ... General > Edit Export Drawings

Creates a new export drawing.

Right-Click Menu – Lets you open, rename, or delete the selected export drawing or create a new export drawing.

Rename

Permissions ... General > Edit Export Drawings

Lets you rename the selected export drawing.

Delete

Permissions ... General > Edit Export Drawings

Deletes the selected export drawing.

Hull Page

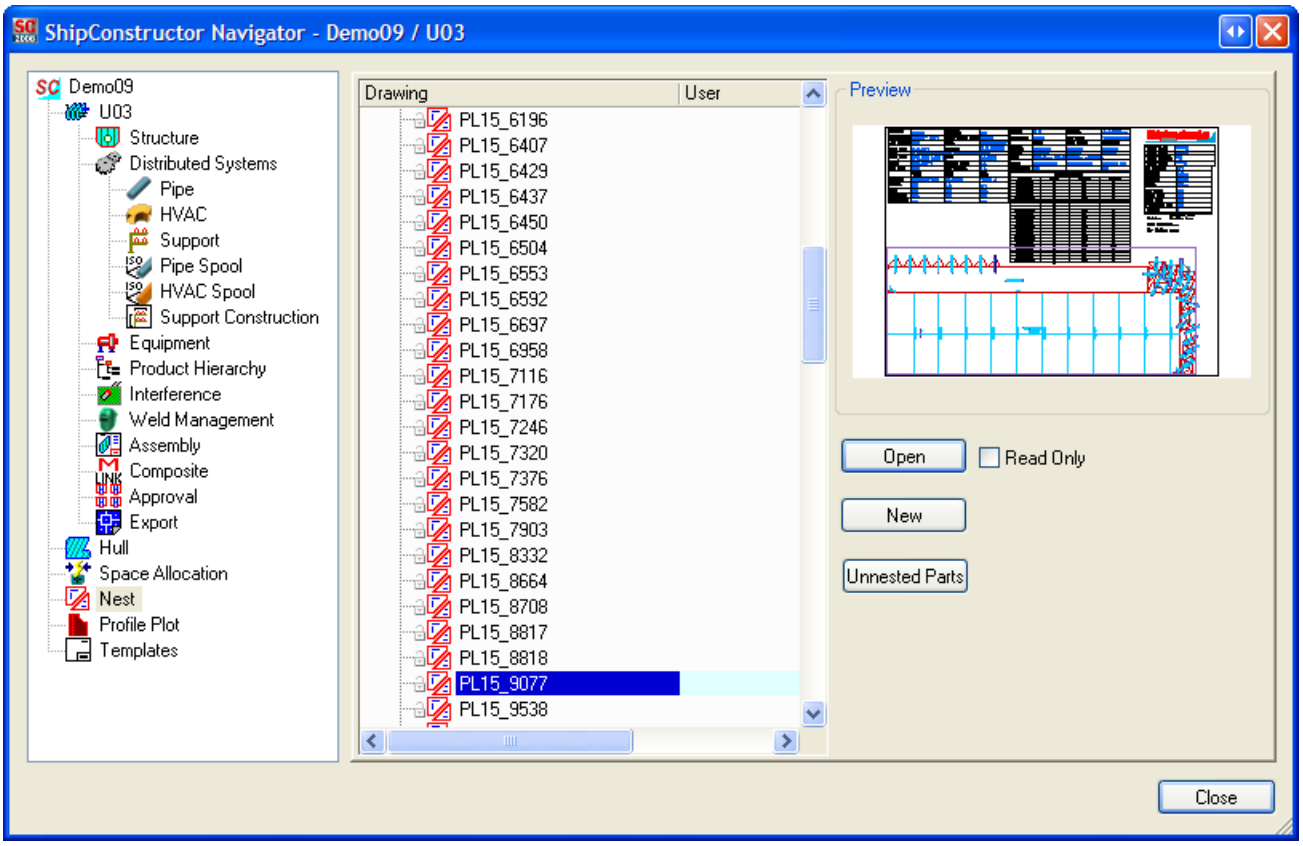
The Hull page lets you create, open, and modify hull drawings. See the Hull manual for details.

Space Allocation Page

The Space Allocation page lets you create, open, and modify space allocation drawings. See the Space Allocation manual for details.

Nest Page

The Nest page lets you create, open, and delete nest drawings. See Plate Nesting (page 254).



Open

Permissions ... None

Opens the selected nest drawing.

New

Permissions ... Plate Nest > Edit Plate Nests

Creates a new nest drawing.

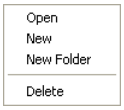
Delete

Permissions ... Plate Nest > Edit Plate Nests

Deletes the selected nest drawing or folder.

Unnested Parts
See SC Plate Nest > Part > List Un-nested Parts (page 518).

Right-Click Menu – Lets you open or delete the selected nest drawing, create a new folder within the nest drawing list, or create a new nest drawing.



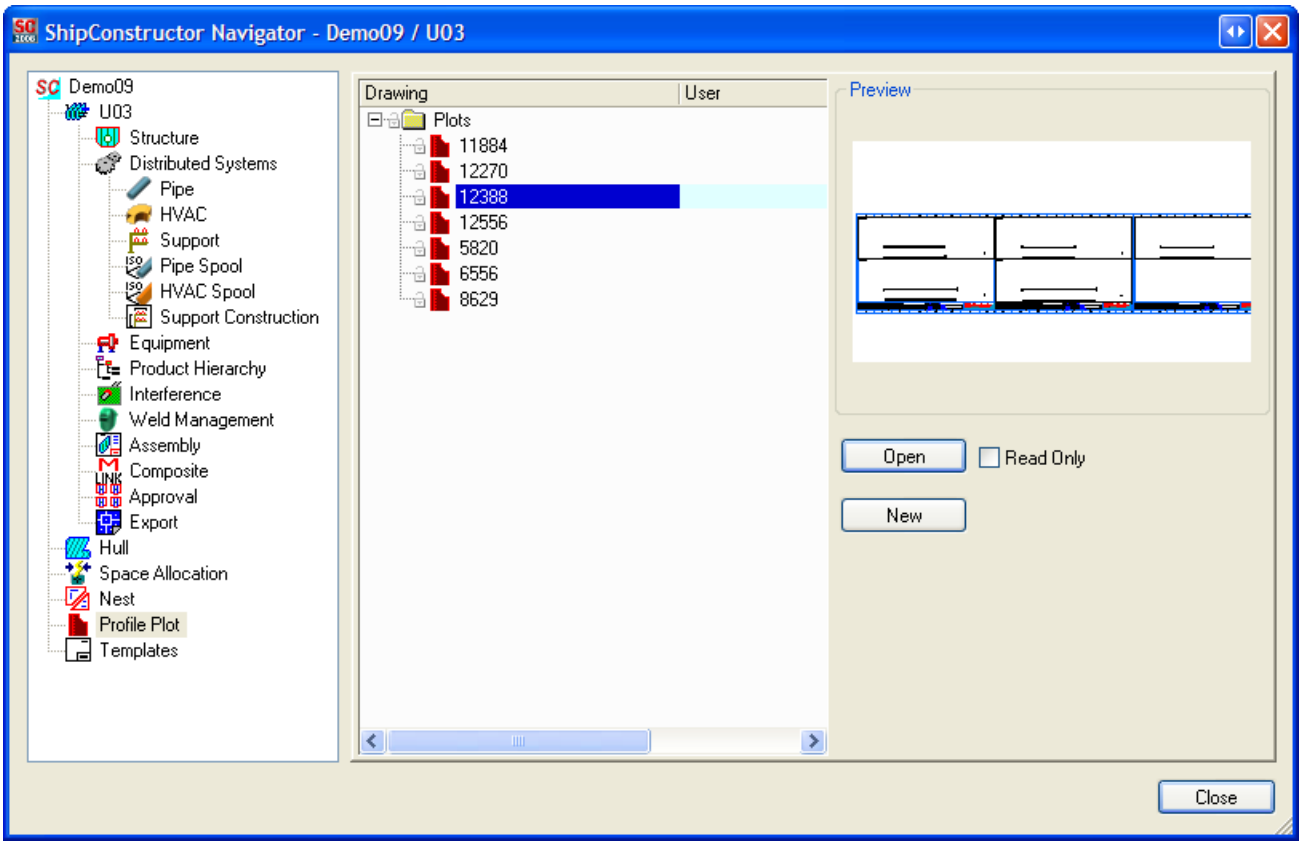
New Folder

Permissions ... None

Creates a new folder within the selected folder in the nest drawing list.

Profile Plot Page

The Profile Plot page lets you create and open profile plot drawings. See Profile Plots (page 295).



Open

Permissions ... None

Opens the selected profile plot drawing.

New

Permissions ... Structure > Edit Profile Plots

Creates a new profile plot drawing.

Right-Click Menu – Lets you open, rename, or delete the selected profile plot drawing or create a new profile plot drawing.

Rename

Permissions ... Structure > Edit Profile Plots

Lets you rename the selected profile plot drawing.

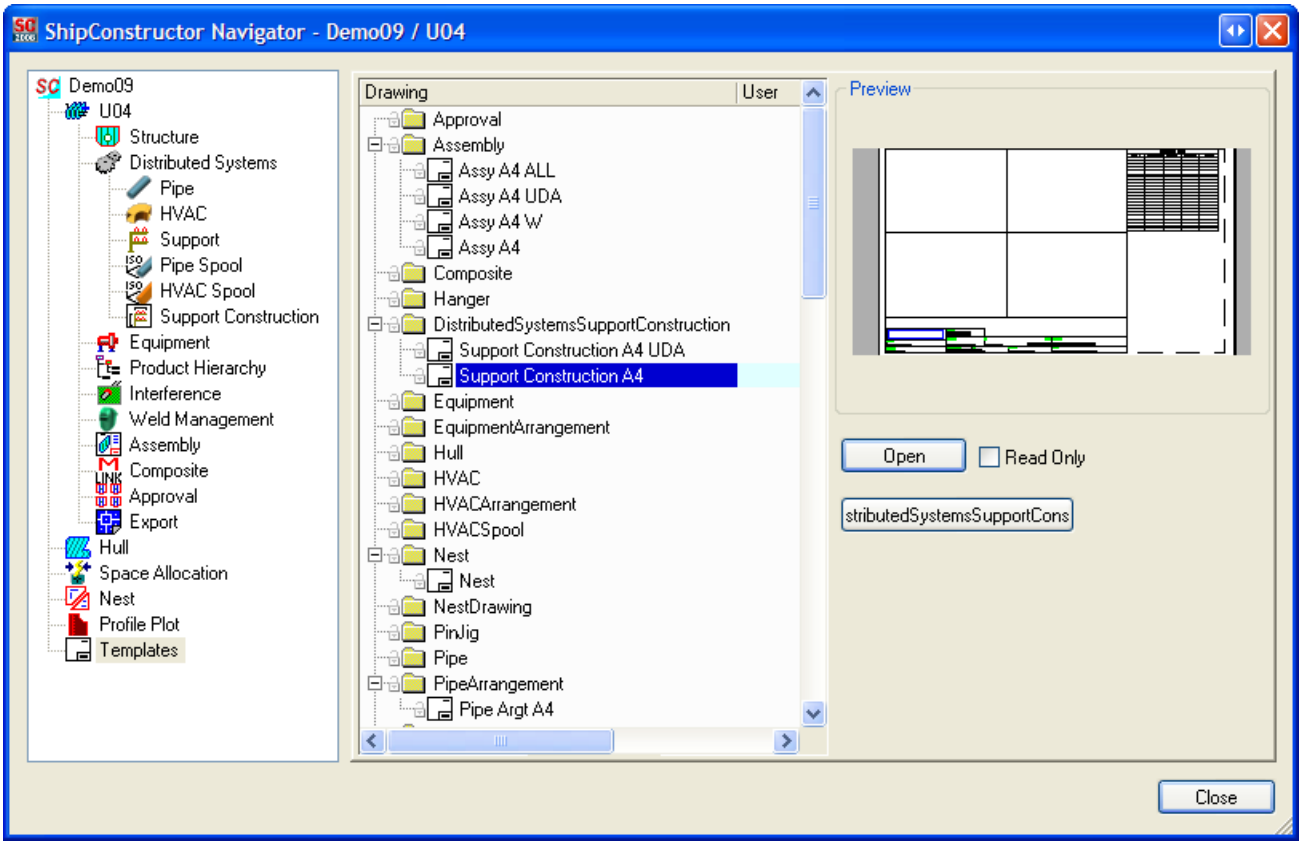
Delete

Permissions ... Structure > Edit Profile Plots

Deletes the selected profile plot drawing.

Templates Page

The Templates page lets you create and open various types of template drawings.



Open

Permissions ... None

Opens the selected template drawing.

New [Template Name]

Permissions ... General > Edit Template Drawings

Creates a new template drawing. The name of the button and the type of template drawing created is based on the selected folder in the Drawing List.

Right-Click Menu – Lets you open, rename, or delete the selected template drawing or create a new template drawing.

Rename

Permissions ... General > Edit Template Drawings

Lets you rename the selected template drawing.

Delete

Permissions ... General > Edit Template Drawings

Deletes the selected template drawing.

ShipConstructor > Manager

Button .....
Ribbon ..... ShipConstructor tab > Applications panel > Manager
Menu ..... ShipConstructor > Manager
Toolbar.....
Command ..... SCMANAGER
Permissions ... None

Opens the Manager window, letting you set up project settings, project libraries, and user permissions.

ShipConstructor > Project > New Project

See the Project Management manual.

ShipConstructor > Project > Copy Project

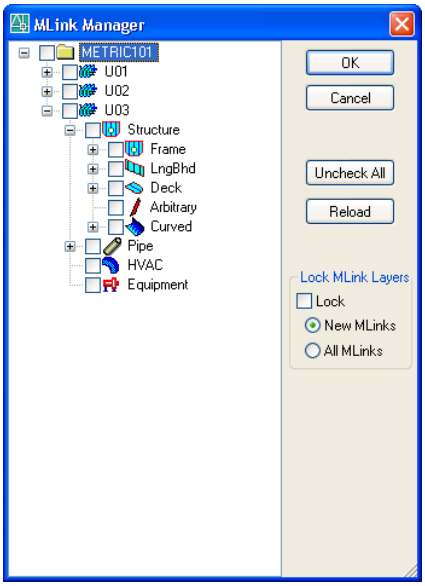
See the Project Management manual.

ShipConstructor > Model Link

Button .....
Ribbon ..... ShipConstructor tab > Navigation panel > Model Link
Menu ..... ShipConstructor > Model Link
Toolbar.....
Command ..... SCMLINK
Permissions ... None
Procedure..... Insert (Link) a Drawing into Another Drawing (page 17)

Lets you insert (link) drawings into the current drawing.

MLink Manager



The MLink Manager lists all of the drawings in the project. You can mlink a drawing into the current drawing by clicking its check box so a checkmark appears. You can remove an mlinked drawing from the current drawing by clicking its check box so it is empty.

Uncheck All – Unchecks all drawings in the list.

Reload – Reloads all mlinked drawings (based on the latest saved version) into the current drawing. For example, if changes have been made to mlinked drawings, you may want to Reload them into the current drawing.

Lock – Locks the layer that the MLink is placed on. Each MLink is placed on its own layer.

New MLinks – Only newly added MLinks will be placed on locked layers

All MLinks – All MLinks are placed on locked layers.

ShipConstructor > Planar Group > Transfer Objects to Group

Button .....
Ribbon ..... Structure Modeling tab > Utilities > Transfer Objects to Group
Menu ..... ShipConstructor > Planar Group > Transfer Objects to Group
Toolbar..... None
Command ..... SCTRANSFER
Permissions ... Structure > Modeling
Procedure..... Transfer Objects from One Planar Group Drawing to Another (page 133)
..... Transfer Objects from a Unit Drawing to a Planar Group Drawing (page 134)

Lets you transfer objects from a unit drawing or a hull drawing to a planar group drawing.

ShipConstructor > Penetration Manager

See the Pipe manual.

ShipConstructor > Check > Check Project

Button .....
Ribbon ..... None
Menu ..... ShipConstructor > Check > Check Project
Toolbar..... None
Command ..... SCCHECKPROJECT
Permissions ... Structure > Modeling
Procedure..... Check a Unit (page 211)

Runs the Check Unit command on all units in the project. The option to repair errors or just log errors is displayed before the check is run. (See Procedure: Check a Unit).

Check Project will also check all template drawings. Repair Errors will confirm that the registered files exist and that all files found of those types are registered.

ShipConstructor > Check > Check Unit

Button .....
Ribbon ..... None
Menu ..... ShipConstructor > Check > Check Unit
Toolbar..... None
Command ..... SCCHECKUNIT
Permissions ... Structure > Modeling
Procedure..... Check a Unit (page 211)

Checks the 3D Unit drawing and all planar group model drawings within the unit. An option to repair errors or log errors is displayed before the check is run. If Repair Errors is not selected, any modifications listed below will not be performed. Check Unit will perform the following checks:

- Makes sure the 3D Unit drawing file exists. If the 3D Unit drawing does not exist, then an empty 3D Unit drawing will be created.

- Checks to see if ShipConstructor specific drawing information exists in drawing. If the unit drawing does not have this drawing information or it is corrupt, ShipConstructor will create or fix the drawing information.
- Calls the Planar Group Check for all planar group model drawings in the unit
- Checks that all markings lie within the boundaries of the part that they are associated with. If any part of a mark symbol or text lies outside the part boundary, it will be noted in the generated log file.
- Checks that the template drawings for Assembly, Pipe/HVAC/Equipment Arrangement, and Pipe/HVAC Spool drawings exist and are registered to the project. Repair Errors will register any files not already registered and missing drawing files will be removed.

Note: Unlike ShipConstructor 2005, checking a unit does not xref all the planar group model drawings into the 3D Unit drawing.

ShipConstructor > Check > Check Templates

Button .....
Ribbon ..... None
Menu ..... ShipConstructor > Check > Check Templates
Toolbar..... None
Command ..... SCCHECKTEMPLATES
Permissions ... Structure > Modeling

Checks that all registered drawings exist, and that all files in the Template folders are registered. Repair Errors will create missing drawings, or register un-registered drawings.

ShipConstructor > Check > Fix Duplicate Drawing Names

Button ..... None
Ribbon ..... None
Menu ..... ShipConstructor > Check > Fix duplicate production drawings
Toolbar..... None
Command ..... SCFIXDUPLICATEPRODDWGS
Permissions ... None

When updating a project from before SC2012R2 you may see production drawings in the project with a $ Duplicate$1, $Duplicate$2, etc. This is because there were duplicate record names in the database. If this is the case, run this command to fix those filenames.

Running this command multiple times has no ill effects.

ShipConstructor > Update Model and System Drawings

Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities > Update Model and System Drawings
Menu ..... ShipConstructor > Update Model and System Drawings
Toolbar..... None
Command ..... SCUPDATEMODELANDSYSTEMDRAWINGSQUICK
Permissions ... None
Procedure..... Update Model and System Drawings (page 212)

Recreates or updates selected model and system drawings from the database.

ShipConstructor > Update Production Drawings

Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities > Update Production Drawings
Menu ..... ShipConstructor > Update Production Drawings
Toolbar..... None
Command ..... SCUPDATEPRODDWGS
Permissions ... None
Procedure..... Update Production Drawings

Updates existing production drawings.

ShipConstructor > Product Hierarchy

Button .....
Ribbon ..... ShipConstructor tab > Navigation panel > Product Hierarchy
Menu ..... ShipConstructor > Product Hierarchy
Toolbar..... Assembly >
Command ..... SCPRODUCTHIERARCHY
Permissions ... None
Procedure..... Product Hierarchies (page 108)

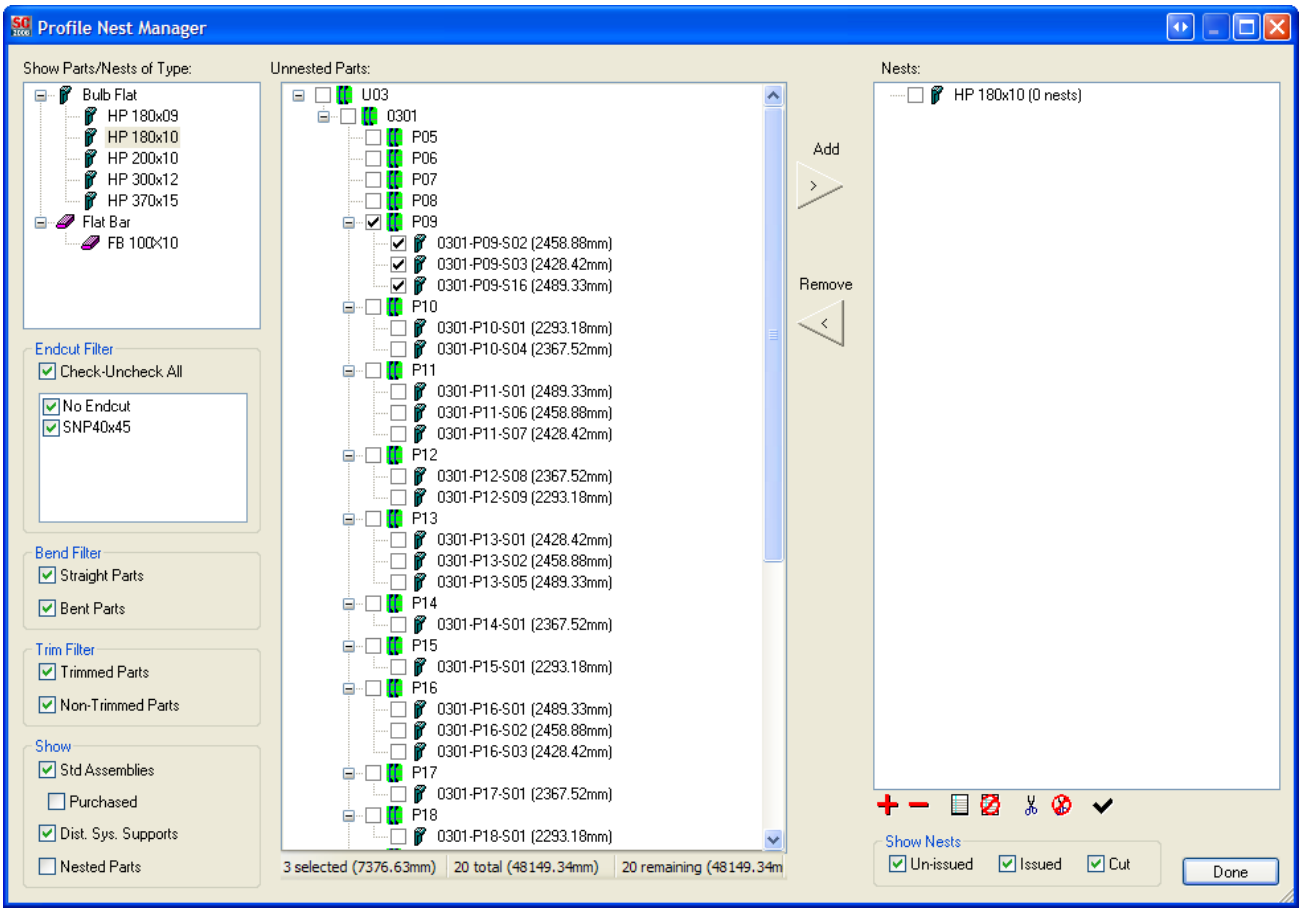
See SC Product Hierarchy > Develop Product Hierarchy (page 508)

ShipConstructor > Profile Nesting

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Nest > Profile Nesting
Menu ..... ShipConstructor > Profile Nesting
Toolbar..... Nest >
Command ..... SCPROFILENEST
Permissions ... Structure > Profile Nest > Profile Nests - Edit
Procedure..... Profile Nesting (page 305)

Opens the Profile Nest window and Profile Nest Manager window, letting you nest parts on profile stocks.

Profile Nest Manager



Show Parts/Nests of Type – Lists the profile stocks used in the unit. Select a profile stock to display a list of parts (under Unnested Parts) and a list of nests (under Nests) for that stock.

Endcut Filter – Filters the Unnested Parts list based on endcut type. Only parts having the selected endcut types (at either end of the part) are displayed. You can also select No Endcuts to display parts without endcuts.

Bend Filter – Filters the Unnested Parts list to display Straight Parts, Bent Parts, and Curved Parts.

Trim Filter – Filters the Unnested Parts list to display Trimmed Parts and Non-Trimmed Parts.

Standard Assemblies – If checked, the filter will display assemblies from standard assemblies, and their parts.

Purchased – If checked, Standard Assemblies tagged as “Purchased” will also be displayed.

Dist. Sys. Supports – If checked, Distributed Systems Supports are displayed.

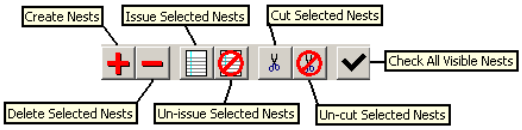
Nested Parts – Includes nested parts in the Unnested Parts list.

Unnested Parts – Lists unnested parts that use the selected profile stock (under Show Parts/Nests of Type) and lets you add parts to nests.

Add – Adds the parts selected under Unnested Parts to the nests selected under Nests. If no nest is selected, ShipConstructor tries to create nests for the selected parts.

Remove – Unnests the parts selected under Nests or unnests all parts from nest selected under Nests.

Nests – Lists the nests for the selected profile stock (under Show Parts/Nests of Type), and the parts within each nest.



Create Nests – Creates an empty profile nest.

Delete Selected Nests – Deletes the selected profile nests and unnests any parts within them.

Issue Selected Nests – Marks the selected profile nests as issued, indicating that it is available for production to cut.

Un-issue Selected Nests – Removes the selected profile nests' issued status.

Cut Selected Nests – Marks the selected profile nests as cut, indicating that the physical plates have been cut.

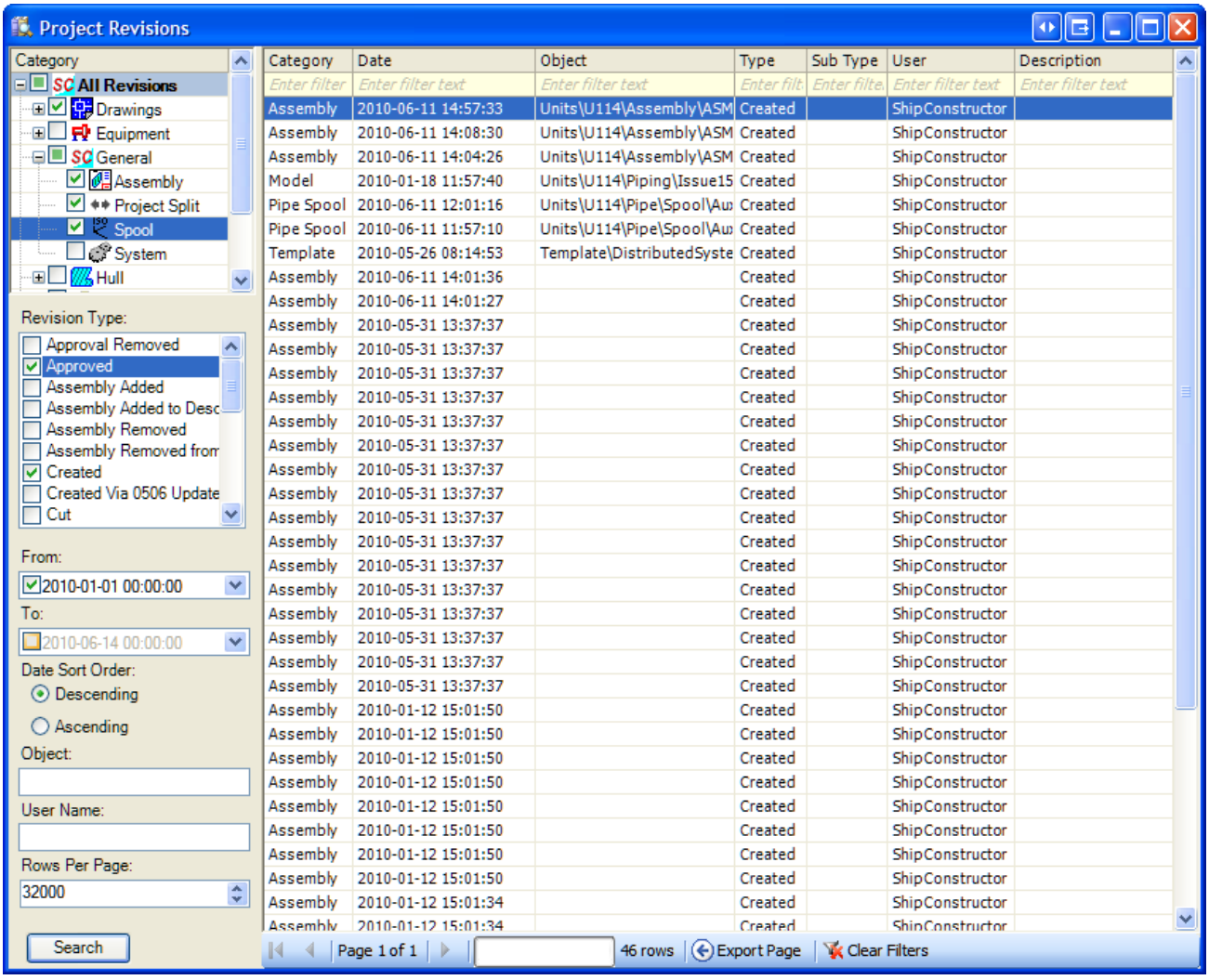
Un-cut Selected Nests – Removes the selected profile nests' cut status.

Check All Visible Nests – Checks all nests (listed under Nests) to see if any parts have been changed, that the parts still fit on the nests and their remnants, and if the stock of any of the parts has been changed. If any part properties have changed, ShipConstructor displays a text file containing a list of the changes.

Show Nests – Filters the Nests list to display UnIssued nests, Issued nests, and Cut nests.

ShipConstructor > Revisions

Button .....
Ribbon ..... None
Menu ..... ShipConstructor > Revisions
Toolbar..... None
Command ..... SCREVISIONS
Permissions ... None
Procedure..... See below



This window queries revisions in the current project. Each revision has the following value:

By default, selecting the **Search** button without changing any options shows all revisions to the project, but the number of revisions can be quite large. Various options are given for restricting the number of retrieved revisions:

- Category – The category of revisions (Category column) from different areas of ShipConstructor. At least one category needs to be selected.
- Revision Type – The type of revision (Type column). At least one type needs to be selected. Right-click options allow you to select all or select none.
- From and To – Optional – Provides a starting and ending date for revision date (Date column). To provide a date, select the check box and select the date area.
- Date Sort Order – Determines the date sort direction

  Descending (present to past) order.

  Ascending (past to present) order.
- Object – Optional – The object the revision pertains to.
- User Name – Optional – The username of the person making the change.
- Rows Per Page – Determines how many revisions to retrieve per page. Revisions are returned in pages since the number of revisions can be quite large. If the number of revisions exceeds the rows per page, the remaining revisions will be available in successive pages.
- Search – Runs the query to fetch rows of revisions.

Other options:

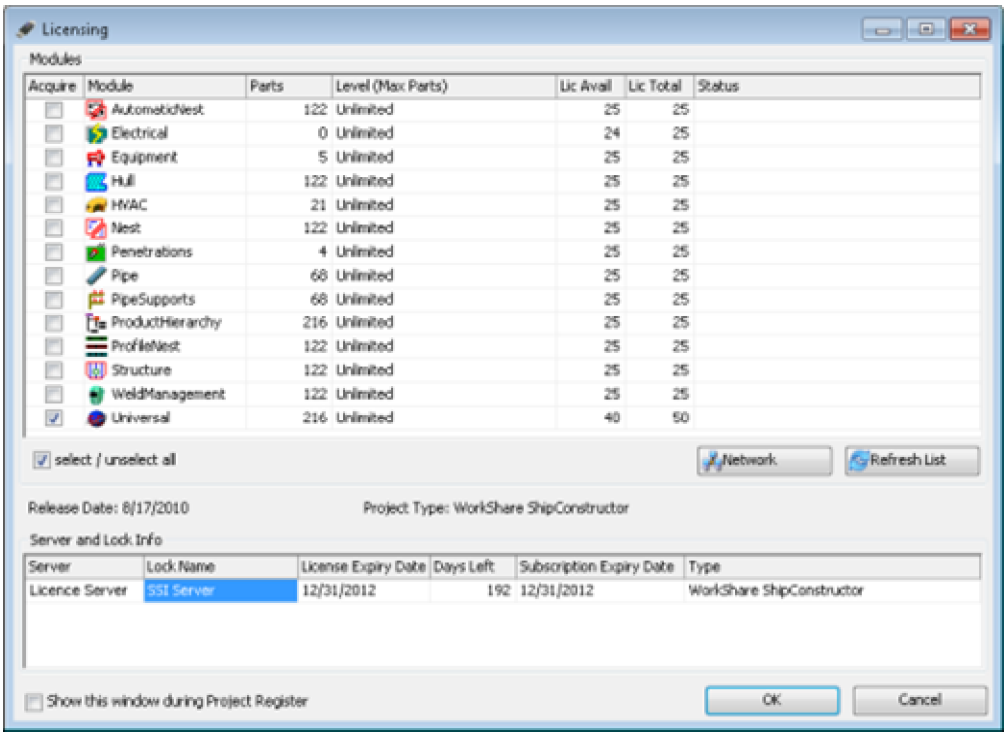
- Filter Row – After rows are fetched, further filtering is performed in the filter row.
- Export Page – A page can be exported to a file in these formats: Text, Microsoft Excel™, or HTML.
- Clear Filters – Clears the filter row.
- Row right-click – Allows you to select search values using values in the currently selected row.

ShipConstructor > Licensing

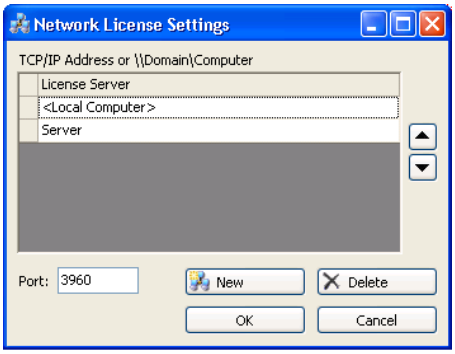
Button .....
Ribbon ..... ShipConstructor tab > Applications > Licensing
Menu ..... ShipConstructor > Licensing
Toolbar..... None
Command ..... SCLICENSE
Permissions ... None
Procedure..... Get Licenses (page 12)

Lets you obtain and change licenses for individual ShipConstructor modules.

Licensing Window



Network – Opens the Network License Settings window, letting you set up a list of computers ShipConstructor searches for licenses. ShipConstructor searches for licenses from the servers in the order they are listed in the Network License Settings window.



- LicenseServer – The list of computers that have network locks attached to them. Any computer with ShipConstructor installed can be a license server. You can specify computers using TCP/IP address, computer name, or \\DOMAIN\computer. The <Local Computer> address is equivalent to IP: 127.0.0.1. Only network locks will allow licenses to be used across a network.
- New – Adds a new license server to the list.
- Delete – Deletes the selected license server from the list. The list must have at least one computer in it. If you try to delete all license servers, the <Local Computer> item is added automatically.
- Move Up – Moves the selected server up in the list.
- Move Down – Moves the selected server down in the list.
- Port – The default port is 3960. You should not need to change this value unless there is a port conflict with another program. If you change the port number, you must also change the port number on all license servers.
- OK – Saves the list and refreshes the modules list in the Licensing window.

Refresh List – Refreshes the module list. This option is useful when locks are moved or license servers are added. If new locks are attached they will be recognized and loaded.

Module – Shows which modules are available (based on available locks). The grayed out modules are not available. Use the check boxes to obtain a license for certain modules.

Parts – The number of parts used in the current project for each module.

Level (Max Parts) – The license level and the maximum number of parts you can have in a project for each module. If you have more than one level available or different options then you will see the ellipsis button. Clicking this button will show a table of the available levels.

Level 5 (3750 Structure Parts)

Level	Max Parts	Lic Avail	Lic Total	Subscription Date	Split & Merge	Status
Level 5	3750	2	2	6/25/2010	Yes	

If a row is disabled then that license level is not selectable and the reason is displayed in the status column.

- For Structure, ManualNest, AutomaticNest, and ProfileNest, the number of structure parts per level is 750. For example, Level 3 allows you to work on projects with a maximum of 2,250 structural parts.
- For Product Hierarchy, the number of parts per level is 1,950. For example, Level 3 allows you to work on projects with a maximum of 5,850 total parts.
- For Pipe, and DistSystemSupport the number of pipe parts per level is 400. For example, Level 3 allows you to work on projects with a maximum of 1,200 pipe parts.
- For Equipment, the number of parts per level is 150. For example, Level 3 allows you to work on projects with a maximum of 450 equipment parts.
- For HVAC, the number of parts per level is 400. For example, Level 3 allows you to work on projects with a maximum of 1,200 HVAC parts.
- For Electrical, the number of cable support parts per level is 100. For example, Level 3 allows you to work on projects with a maximum of 300 cable support parts.

- For Penetrations, the number of penetrations per level is 150. For example, Level 3 allows you to work on projects with a maximum of 450 penetrations.

Lic Avail – The number of licenses available (that is, the total number of licenses minus the number of licenses currently in use).

Lic Total – The total number of licenses.
Status – Shows information about a license. The following is a list of statuses and what they mean.

Status	Action
In Use	All the licenses for this module are used up by other users. You cannot get a license until another user gives up their license or you force release that license using LicenseMonitor.
Old version	The version in the lock is lower than the software needs. If your Subscription is up-to-date, contact your dealer.
License is not WorkShare	If the project is a WorkShare (formerly Project Split & Merge) project (project has a split project) only licenses which support WorkShare will be able to be used on this project.
Subscription Expired!	The Subscription expiry date is older than the release date of this version of the software. Contact your dealer in order to renew your Subscription.
Part Count Exceeded!	The License has a part count limit that is exceeded by the part count in the project. Contact your dealer to upgrade the level of the license.

Release Date – The date that the software was released.

Project Type – Indicates if the project is a WorkShare project or not.

Server – The name of the license server computer

Lock Name – The name of the lock. This usually contains the serial number as well.

Expiry Date – The date when the lock expires.

Days Left – The number of days until the lock expires.

Subscription Expiry Date – The subscription expiry date stored in the lock. You are permitted to install software that has release dates prior to this date.

Type – Indicates if the lock is an ordinary ShipConstructor lock or a WorkShare (formerly Project Split & Merge) lock.

Show this window during Project Register – Displays the Licensing window whenever you register a project. Turn this option off if you always get the same licenses or only use a local lock. (You can open the Licensing window using ShipConstructor > Licensing).

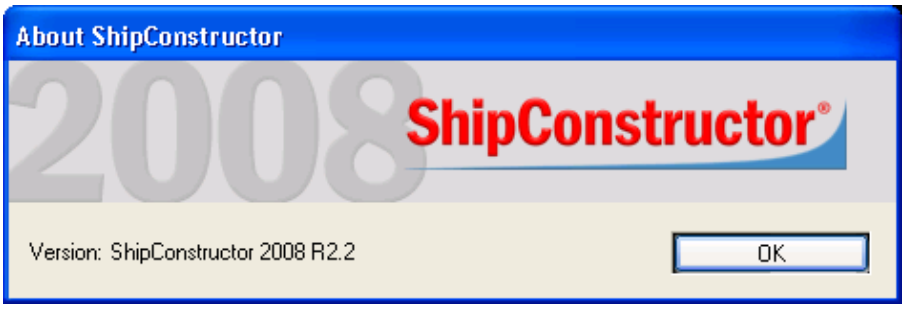
OK – Gets the selected licenses from the license servers.

ShipConstructor > ShipConstructor Help

Lets you access the ShipConstructor online help and individual manuals (in PDF format).

ShipConstructor > About ShipConstructor

Displays the ShipConstructor version numbers.



Version – The version of the ShipConstructor program.

SC Structure Menu

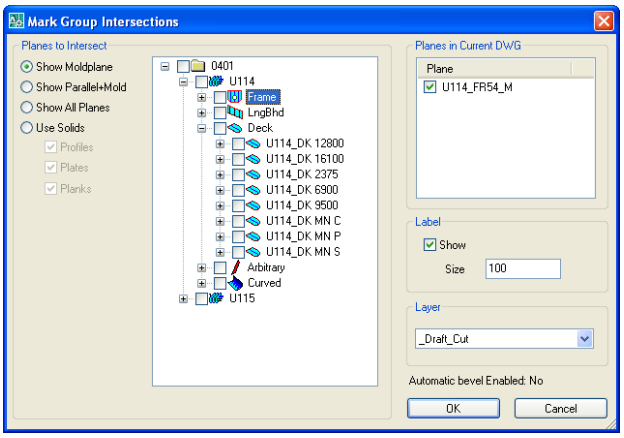
The SC Structure menu is available within planar group and curved plate drawings.

SC Structure > Mark Group Intersections

Button .....
Ribbon ..... Structure Modeling tab > Utilities panel > Mark Group Intersections
Menu ..... SC Structure > Mark Group Intersections
Toolbar..... Structure >
Command ..... SCMARK
Permissions ... None
Procedure..... Mark Group Intersections (page 157)

Marks the intersections between a specified planar group plane and a plane in the current drawing. This command will generate PlanarGroupPlane construction lines. They cannot be moved, only extended. This prevents mistakes of where planes are located. You can explode or copy the drawing to create lines that can be modified.

Mark Group Intersections Window



The current drawing is indicated by the icon.

The active UCS is automatically selected under Current Dwg UCS.

Note: Some of the listed planes may not intersect with planes in the current drawing (for example, if their XY planes are parallel). After you click OK, ShipConstructor will let you view a log file listing planes that do not intersect each other.

Show Moldplane – Displays only the moldplanes from the project.

Show Parallel+Mold – Displays only the moldplanes and planes parallel to the moldplanes for each planar group drawing.

Show All Planes – Displays all planes for the whole project.

Use Solids – Fills the tree with planar groups and intersects the parts in the checked planar groups with the Planes in Current DWG.

Profiles – Intersects stiffeners and faceplates in the checked planar groups with the Planes in Current DWG.

Plates – Intersects plates in the checked planar groups with the Planes in Current DWG.

Planes in Current DWG – Lists the UCS in the current planar group drawing. By default, the active plane is pre-selected in the list.

Label – Labels the line with the name of the group. The Size field lets you set the size of the text label.

Layer – Allows you to select the layer the PlanarGroupPlane construction line will be placed on.

SC Structure > Show / Hide Options

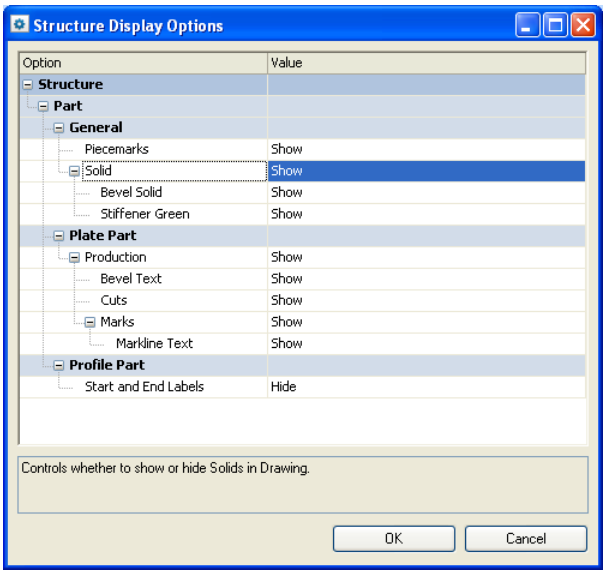
Command ..... SCSTRUCTDISPLAY
Permissions ... None

This command is no longer supported. SCDWGOPTIONSSTRUCT should be used instead.

Button .....
Ribbon ..... Structure Modeling tab > Drawing Options panel > Structure Drawing Options
Menu ..... SC Structure > Structure Drawing Options

Toolbar..... Structure Display >
Command ..... SCDWGOPTIONSSTRUCT
Permissions ... None

Opens the Structural Display Options window.



General

Piecemarks – Controls whether to display part's piecemarks in the current drawing.

Solids – Controls whether to display part solids in the current drawing.

Bevel Solid – Controls whether to display bevel solids if displaying part solids in the current drawing.

Stiffener Green – Controls whether to display stiffener green if displaying part solids in the current drawing.

Plate Part

Production – Controls whether to display plate production objects in the current drawing.

Bevel Text – Controls whether to display bevel text if displaying production in the current drawing.

Cuts – Controls whether to display plate cut objects if displaying production in the current drawing.

Marks – Controls whether to display plate mark objects if displaying production in the current drawing.

Markline Text – Controls whether to display markline text if displaying marks in the current drawing.

Stiffener

Start and End Labels – Controls whether to display start and end labels for stiffeners in the current drawing.

The following toolbar buttons provide quick access to specific options above.

Toggle Show Production

Toggle Show Solid

Toggle Show Draft

The following commands toggle display options for the current drawing.

Production

Button .....
Ribbon ..... Structure Modeling tab > Drawing Options panel > Toggle Show Production

Toolbar..... Structure Display >
Command ..... SCSTRUCTDISPLAYTOGGLEPROD
Permissions ... None

Toggles the display of production objects in the current drawing.

Command ..... SCDWGOPTIONSSTRUCTSHOWPROD
Permissions ... None

Prompts the user to hide or show production objects in the current drawing.

Solids

Button .....
Ribbon ..... Structure Modeling tab > Drawing Options panel > Toggle Show Solid

Toolbar..... Structure Display >
Command ..... SCSTRUCTDISPLAYTOGGLESLD
Permissions ... None

Toggles the display of part solids in the current drawing.

Command ..... SCDWGOPTIONSSTRUCTSHOWSLD
Permissions ... None

Prompts the user to hide or show part solids in the current drawing.

Draft Layers

Button .....
Ribbon ..... Structure Modeling tab > Drawing Options panel > Toggle Show Draft

Toolbar..... Structure Display >
Command ..... SCSTRUCTDISPLAYTOGGLEDRFT
Permissions ... None

Toggles the display of draft objects in the current drawing.

Command ..... SCDWGOPTIONSSTRUCTSHOWDRFT
Permissions ... None

Prompts the user to hide or show draft objects in the current drawing.

Piecemarks

Command ..... SCSTRUCTDISPLAYTOGGLEPMARK
Permissions ... None

This command is no longer supported. SCDWGOPTIONSSTRUCTSHOWPMARK should be used instead.

Command ..... SCSTRUCTDISPLAYSHOWPMARK
Permissions ... None

Prompts the user to hide or show piecemarks in the current drawing.

Part Cuts

Command ..... SCSTRUCTDISPLAYTOGGLECUT
Permissions ... None

This command is no longer supported. SCDWGOPTIONSSTRUCTSHOWCUT should be used instead.

Command ..... SCSTRUCTDISPLAYSHOWCUT
Permissions ... None

Prompts the user to hide or show part cuts in the current drawing.

Part Markings

Command ..... SCSTRUCTDISPLAYTOGGLEMARK
Permissions ... None

This command is no longer supported. SCDWGOPTIONSSTRUCTSHOWMARK or SCDWGOPTIONSSTRUCTSHOWMARKTEXT should be used instead.

Command ..... SCDWGOPTIONSSTRUCTSHOWMARK
Permissions ... None

Prompts the user to hide or show part markings in the current drawing.

Command ..... SCDWGOPTIONSSTRUCTSHOWMARKTEXT
Permissions ... None

Prompts the user to hide or show part text markings in the current drawing.

Part Bevels

Command ..... SCSTRUCTDISPLAYTOGGLEBEVEL
Permissions ... None

This command is no longer supported. SCDWGOPTIONSSTRUCTSHOWBEVELSLD or SCDWGOPTIONSSTRUCTSHOWBEVELTEXT should be used instead.

Command ..... SCDWGOPTIONSSTRUCTSHOWBEVELSLD
Permissions ... None

Prompts the user to hide or show part bevel solids in the current drawing.

Command ..... SCDWGOPTIONSSTRUCTSHOWBEVELTEXT
Permissions ... None

Prompts the user to hide or show part bevel text in the current drawing.

Extrusion Greens

Command ..... SCSTRUCTDISPLAYTOGGLEGREEN
Permissions ... None

This command is no longer supported. SCDWGOPTIONSSTRUCTSHOWGREEN should be used instead.

Command ..... SCDWGOPTIONSSTRUCTSHOWGREEN
Permissions ... None

Prompts the user to hide or show extrusion greens in the current drawing.

Extrusion Labels

Command ..... SCSTRUCTDISPLAYTOGGLELABEL
Permissions ... None

This command is no longer supported. SCDWGOPTIONSSTRUCTSHOWLABEL should be used instead.

Command ..... SCDWGOPTIONSSTRUCTSHOWLABEL
Permissions ... None

Prompts the user to hide or show extrusion start and end labels in the current drawing.

SC Structure > Plate > New

Button .....
Ribbon ..... Structure Modeling tab > Plates panel > New Plate
Menu ..... SC Structure > Plate > New

Toolbar..... Structure > Structure Plate Flyout >
Command ..... SCCREATEPLATE
Permissions ... Structure > Modeling
Procedure..... Plate Part Creation (page 158)

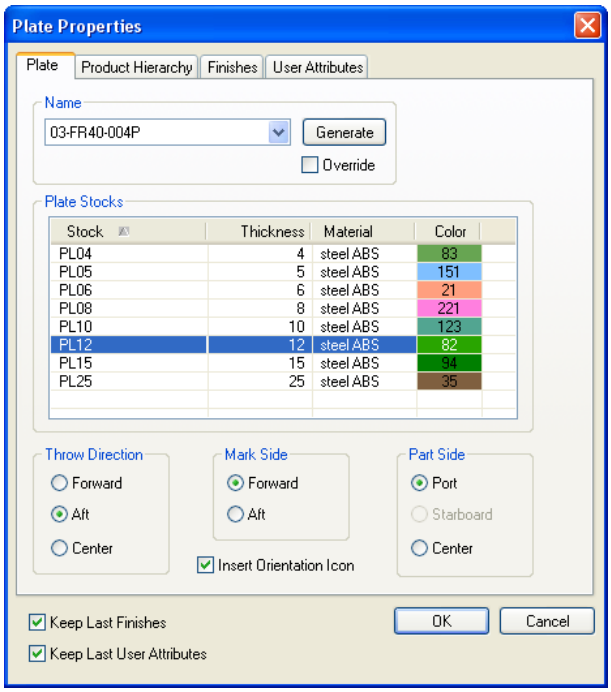
Creates a new plate part.

Plate parts are created by picking a point inside a bounded area of cut lines. Cut lines are lines, arcs, polylines, circles, and construction lines on the _Draft_Cut layer. The create new plate command automatically switches all other layers off so that you can verifying the objects that will be considered for creating a plate. You can alternatively pre-select objects to define a plate. This is useful when your cut line overlaps the intended edge of the part you want to define.

Parts are not added to the database until the drawing is saved.

The Plate Properties window contains the following tabs:

- Plate– Lets you select a plate stock and set other part properties.



Name – The name that the new plate will use. Names can either follow the naming convention or override the naming convention. If you manually enter a name, the name is not following the naming convention and will not be updated when properties of the name change. A drop window lets you select a generated name from a list in case you do not define the parts in the order you want them named.

Generate – Generates a new name for this part using the current naming convention.

Override – Lets you manually create a name for this part, overriding the current naming convention. This prevents the name from changing when a part property that is part of the name convention is changed.

Plate Stocks – The stock that will be used to create the new plate.

Throw Direction – The throw direction of the plate determines which way the thickness of the plate is relative to the mold side of the plate. The production information is shown on the mold side of the plate.

Mark Side – The mark side determines which side of the plate will face up when the part is in the nest drawing. The displayed directions are determined by the orientation of the plate in the model.

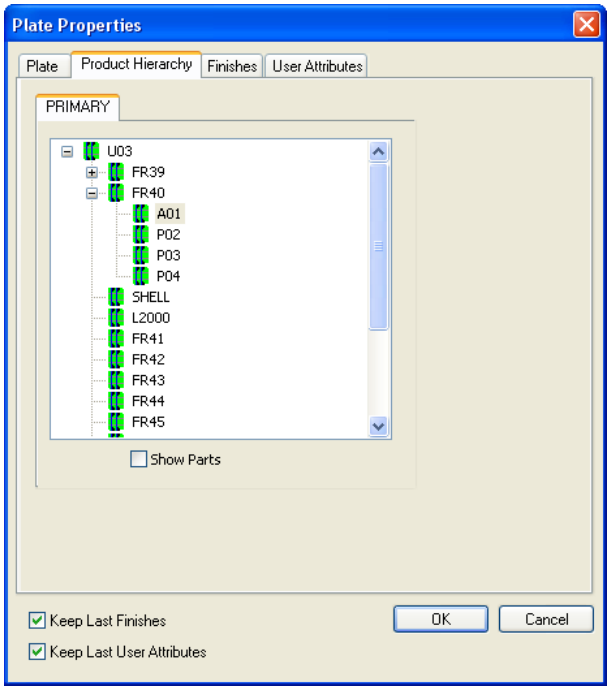
Part Side – The side of the vessel that the part is located. ShipConstructor calculates this automatically based on CG position.

Insert Orientation Icon – An orientation icon is automatically inserted into the part when this is checked. The orientation icon uses the settings from Structure > Plates > Orientation Icon to determine the appearance of the icon. The orientation icon is placed in the lower left area of the part by default. You can move the orientation by selecting the grip point on it. This setting is remembered when creating subsequent plate parts in that drawing.

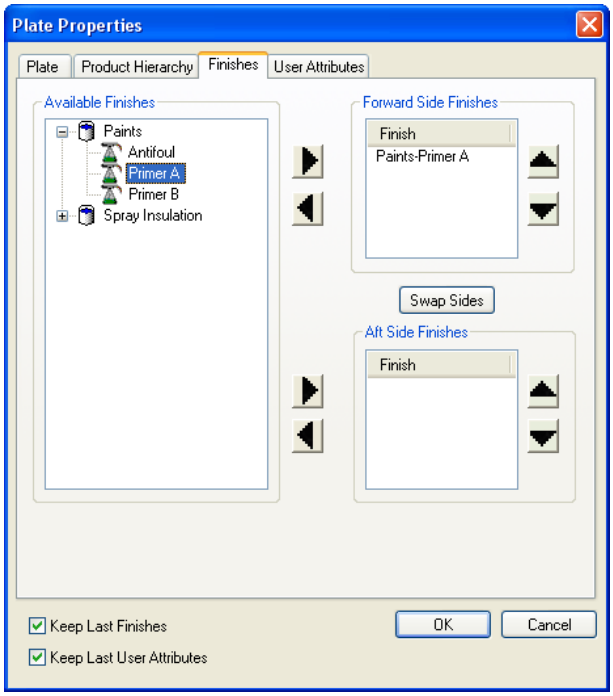
Keep Last Finishes – Adds the finishes selected when creating the last part to this new part.

Keep Last User Attributes – Adds the user attributes selected when creating the last part to this new part.

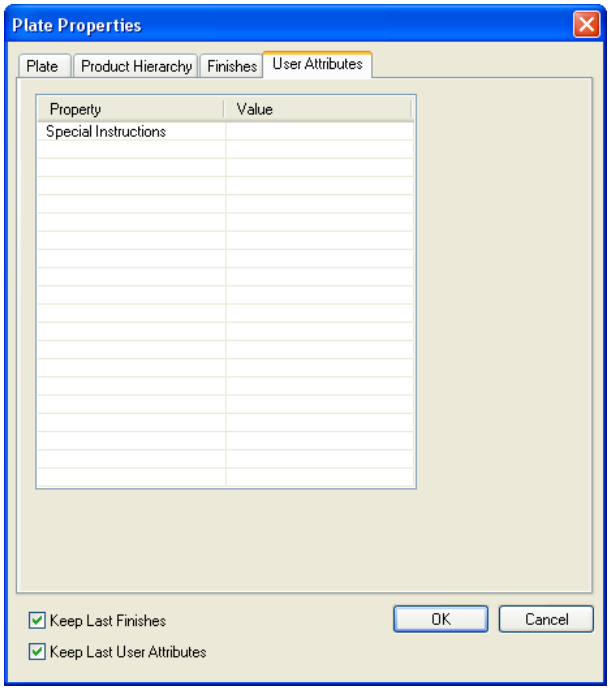
- Product Hierarchy – Lets you assign the part to an assembly. (By default the part is assigned to the default assembly for the planar group.)



- Finishes – Lets you apply different finishes to each side of the plate.



- User Attributes – Lets you set arbitrary attributes for the part (for example, model number). User attributes are displayed with part properties.



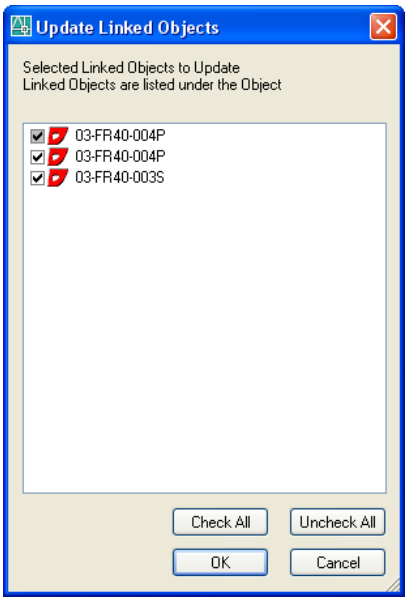
SC Structure > Plate > Edit

Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Edit
Menu ..... SC Structure > Plate > Edit

Toolbar..... Structure > Structure Plate Flyout >
Command ..... SCEDITPLATE
Permissions ... Structure > Modeling
Procedure..... Edit Plate

Edits an existing plate part. This allows modification of the plates properties.

If a plate has related parts (identical or mirrored parts) then you will see the Update Linked Objects window. If you uncheck parts from the list then the checked items will become unrelated to the unchecked ones. Only checked items will be updated. If there is any unchecked item then the part names of the checked items will change.

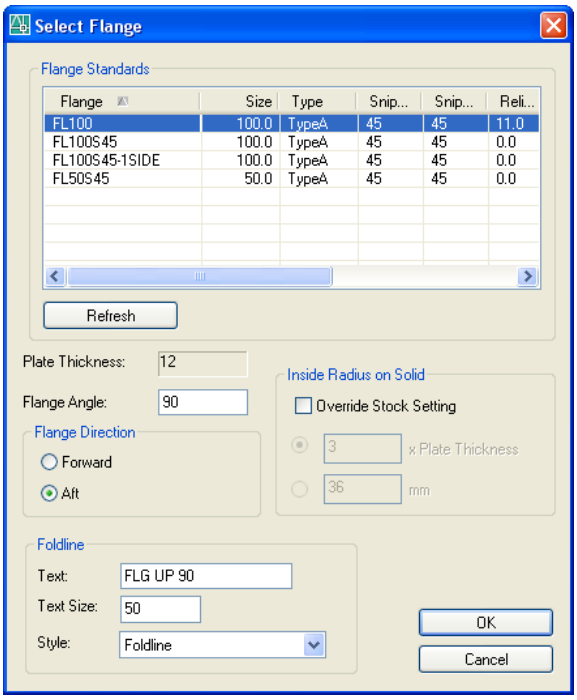


SC Structure > Plate > Add Flange

Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Flange Plate drop-down > Add Flange
Menu ..... SC Structure > Plate > Add Flange

Toolbar..... Structure > Structure Flange Plate Flyout >
Command ..... SCPLATEFLANGE
Permissions ... Structure > Modeling
Procedure..... Plate Flanges (page 163)

Adds a flange to a plate part.



Flange Standards – The list of flange standards. The selected flange standard is used to create the flange. See Create a Flange Standard (page 79) for how to set up flange standards.

Plate Thickness – The thickness of the selected plate.

Flange Angle – The angle to bend the flange relative to the plane of the flat plate.

Direction – The direction to flange the plate. The directions are determined by the orientation of the plate. If the flange direction is in the opposite direction of the mark side of the plate, you will see the following warning.

ShipConstructor
The foldline for this flange will not be marked on the plate's mark side. Switch the plate's mark side to correct this?
Yes
No

Clicking Yes will change the plate's mark side so that this flange foldline is mark on the top side of the plate if it is laying flat.

Flange Text - The text that the function places at the center of the flange line. The default value used for this is stored in Plate Flange Default Text in General > Project Settings

Text Size - Size of the text.

Style - The markline style of the foldline of the flange. The available markline styles listed are set up in the Assign Markline Styles window.

Inside Radius on Solid
The flange is added to the plate as a solid in the final bent condition and as an expanded flange in production. The flange solid is attached to the plate using a radius corner to simulate actual bending. The bending inside radii standards are set up in Flg Inner Rad (X Thick) in the Structure Stock Catalog.

Override Stock Setting – You can override the standard inside radius by checking this box and setting a new inside radius.

Overriden Inside Radius – The custom radius can be entered as a value in the project units or as a multiple of the plate thickness.

SC Structure > Plate > Edit Flange

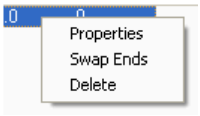
Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Flange Plate drop-down > Edit Flange
Menu ..... SC Structure > Plate > Edit Flange

Toolbar..... Structure > Structure Flange Plate Flyout >
Command ..... SCPLATEEDITFLANGE
Permissions ... Structure > Modeling
Procedure..... Plate Flanges (page 163)

Lets you edit any of a plate's flanges. After you select this command and then click on a plate with a previously defined flange, the Flange List window appears.

From the list, double click the flange you want to edit.

Alternatively, you can also right-click the selected flange. The following menu appears:



Properties – Opens the Select Flange window that you can use to edit the details of the flange.

Swap Ends - Reverses the first and second relief radius and snipe angle for the flange. This will only affect the selected flange.

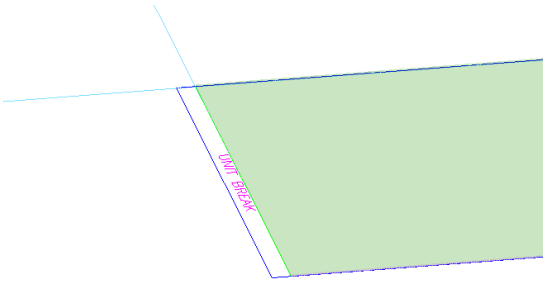
Delete - Deletes the selected flange.

SC Structure > Plate > Add Green

Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Structure Green drop-down > Add Green
Menu ..... SC Structure > Plate > Add Green

Toolbar..... Structure > Structure Green Flyout >
Command ..... SCADDPPGREEN
Permissions ... Structure > Modeling
Procedure..... Plate Green (or Stock) (page 164)

This command adds extra stock to the edge of a plate part. This extra stock piece on the part can be changed or removed. The extra stock is not added to the solid and does not affect the weight or center of gravity of the part. The plate green can only be added to a portion of the outside toolpath of a plate part. When the edge of a plate is moved, the start and end points for the green are recalculated to be the closest points to where they were.

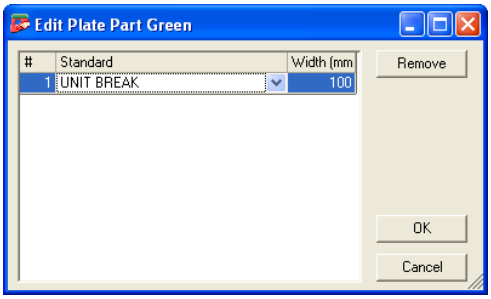


SC Structure > Plate > Edit Green

Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Structure Green drop-down > Edit Green
Menu ..... SC Structure > Plate > Edit Green

Toolbar..... Structure > Structure Green Flyout >
Command ..... SCEDITPPGREEN
Permissions ... Structure > Modeling
Procedure..... Plate Green (or Stock) (page 164)

Lets you change the standard or remove any of the green added to a plate part.



The selected cuts are added to the existing collection and the affected plank parts are updated. Previously identical plank parts within a collection may be renamed if the plank outlines changes.

SC Structure > Plate > Add Objects

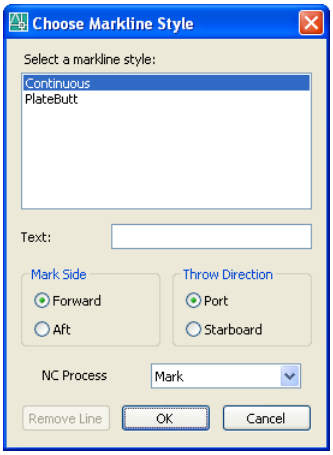
Button .....
Ribbon ..... Structure Modeling tab > Parts panel > Add Objects to Part
Menu ..... SC Structure > Plate > Add Objects

Toolbar..... Structure > Structure Plate Objects Flyout >
Command ..... SCADDOBJECTSTOSTRUCTPART
Permissions ... Structure > Modeling
Procedure..... Add, Remove, and Edit Objects on an Existing Plate Part (page 160)
..... Adding and Removing Cut Lines to a Plank Collection (page 192)

Adds objects to a structure part. The layer of the object being added determines how it is added to the part. Objects on the _Draft_Cut layer are added to the toolpath. Objects on the _Draft_NoProcess or _Draft_Mark layer become marklines on the plate.

Adding Mark Objects

Adding lines, arcs, polylines, circles, construction lines, or ellipses as marks shows the following window.



Markline Style – The list of available styles to use. This list is made from the assigned marklines to General category.

Text – Optional text string to label the markline.

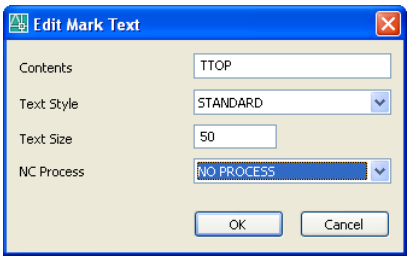
Mark Side – The side of the plate that this markline will be marked.

Throw Direction – The positive side of the markline. This controls the marksymbol's orientation.

NC Process – The NC process of the markline. This overrides the markline style setting for this markline.

Remove Line – Removes the current markline from the plate. Disabled during Add Objects.

Adding Mark Text



Contents – Text string place on the part. The TEXT object's contents are shown but can be changed.

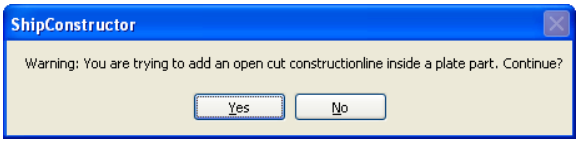
Text Style – The text style to use. This list will contain the list of ShipConstructor text styles (see General > Text Styles (page 335)) and not the AutoCAD text styles in the drawing.

Text Size – the height of the text. The TEXT's height is the initial value.

NC Process – Indicates whether you want to mark the text on the plate or not.

Adding Cut Objects

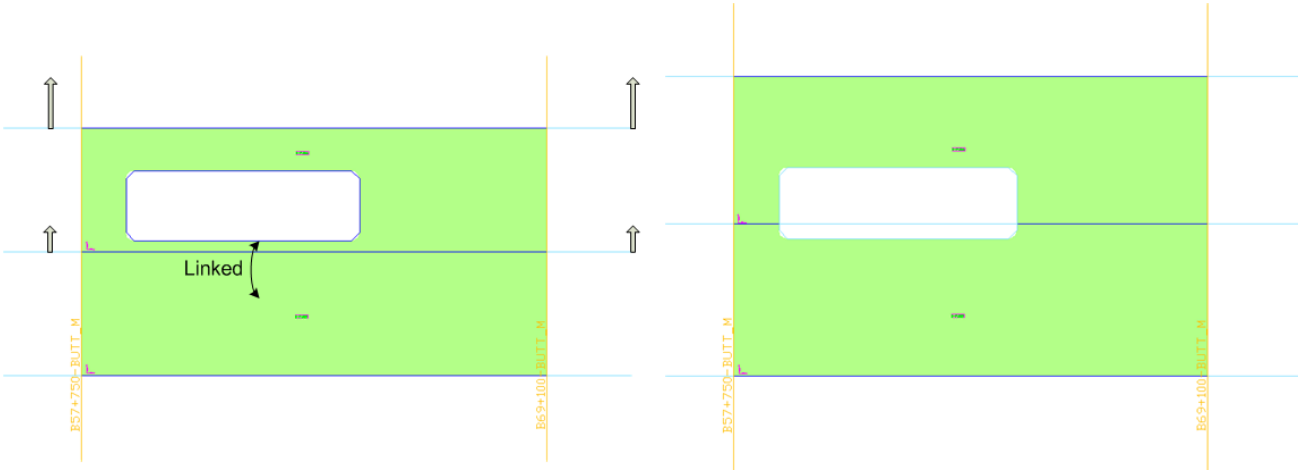
Cut objects can be added to the toolpath. If you attempt to add a cut that is not closed as a hole you will see the following warning.



Choosing Yes will add the object to the plate part but the plate part will not include it in the toolpath.

Cut objects that are not completely inside the part are added automatically even if they do not become part of the toolpath.

You may want to add cut objects to plate parts because other boundaries of the part may move enough to later include the cut object.



By adding the hole polyline to the lower plate, moving the seams (because you decided to use wider plates) automatically trims the bottom plate to the opening.

SC Structure > Plate > Remove Objects

Button .....
Ribbon ..... Structure Modeling tab > Parts panel > Remove Objects from Part
Menu ..... SC Structure > Plate > Remove Objects

Toolbar..... Structure > Structure Plate Objects Flyout >
Command ..... SCREMOVEOBJECTSFROMSTRUCTPART
Permissions ... Structure > Modeling
Procedure..... Add, Remove, and Edit Objects on an Existing Plate Part (page 160)
..... Adding and Removing Cut Lines to a Plank Collection (page 192)

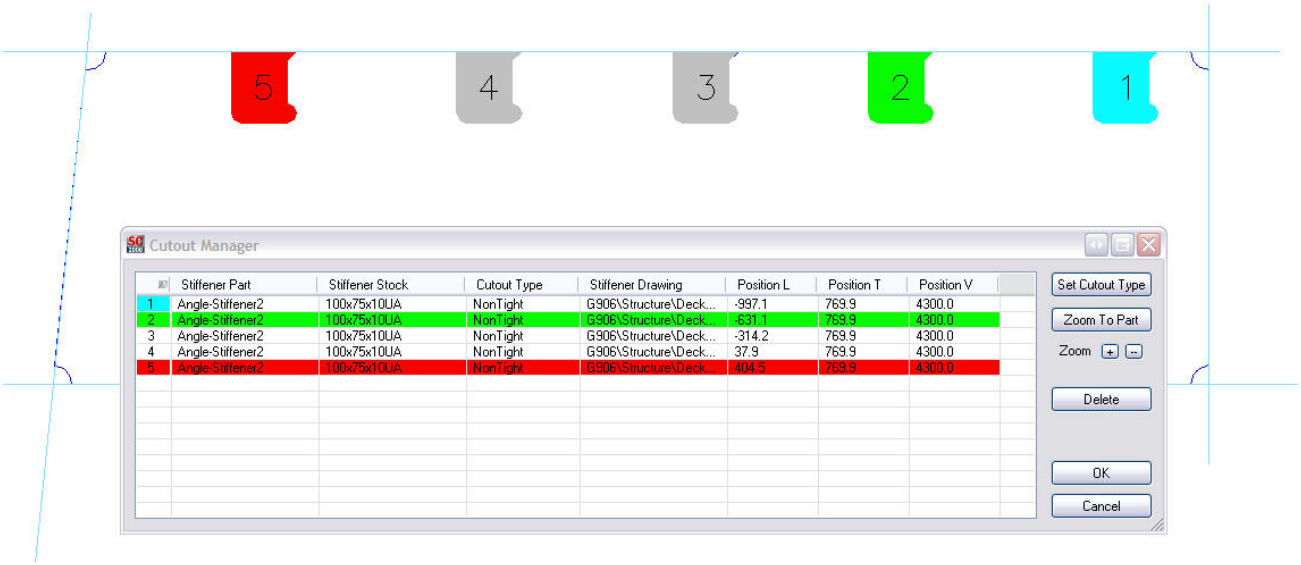
Removes marklines from a structure part. Currently only marklines and mark text can be removed.

SC Structure > Plate > Manage Cutouts...

Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Manage Cutouts
Menu ..... SC Structure > Plate > Manage Cutouts...

Toolbar..... Structure >
Command ..... SCPLATECUTOUT
Permissions ... Structure > Modeling
Procedure..... Plate Cutouts (page 162)

Adds cutouts caused by stiffeners crossing a plate. All the stiffeners, faceplates, and twisted stiffeners in the whole project are considered but those not in the current unit may not be added (they will show up as deleted), depending on the project settings (Structure > Allow Remote Profile Cutouts in the section General > Project Settings (page 327)). If the stiffener crosses the plate at an angle then the cutout is automatically stretched.



The cutouts are temporarily labeled to make it easier to determine the associated one on the part. These labels are color coded to further help:

Green – New Cutout

Red – Deleted Cutout

Yellow – Modified Cutout

Cyan – Selected Cutout

Double-clicking on the cutout in the list will zoom to that cutout in the model.

Stiffener Part – The part name of the stiffener that intersects this plate part.

Stiffener Stock – The stock of the stiffener part

Cutout Type – A drop-down list containing the list of cutout types. See Structure > Structure Stock Catalog > Cutout Shap... (page 345) on how to set up cutout types.

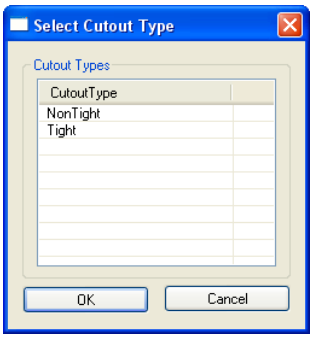
Stiffener Drawing – The drawing that contains the stiffener part.

Position L – Longitudinal location of where the moldline point of the stiffener intersects the plate.

Position T – Transverse location of where the moldline point of the stiffener intersects the plate.

Position V – Vertical location of where the moldline point of the stiffener intersects the plate.

Set Cutout Type – Changes all the selected cutouts to the selected type. This is a fast way to change all the cutouts without having to change them individually.



Zoom to Part – Zooms to the plate part extents.

Zoom + and - – Zooms in and out

Delete – Deletes the selected cutouts from the part. If Manage Cutouts.. is run later, the deleted cutout will reappear. This button is also used to un-delete deleted cutouts.

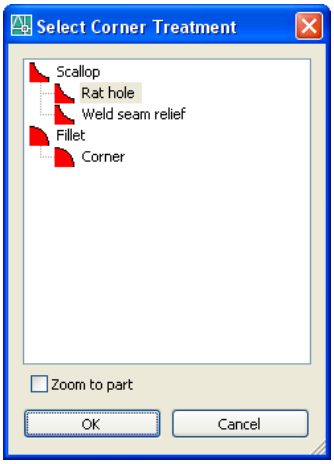
SC Structure > Plate > Corner Treatment > Insert

Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Structure Corner Treatment drop-down > Insert Corner Treatment
Menu ..... SC Structure > Plate > Corner Treatment > Insert

Toolbar..... Structure > Structure Corner Treatment Flyout >
Command ..... SCADDCORNERTREATMENT
Permissions ... Structure > Modeling
Procedure..... Corner Treatments (page 81)

Inserts a corner treatment in a plate part. A corner treatment is a cut detail inserted into the cut toolpath of a plate part. Typically these are rat holes or weld seam reliefs. The corner treatment references the intersection between two construction lines. If the construction lines move, then the corner treatment moves.

Corner treatment standards are set up in Structure > Plates > Corner Treatments.



Zoom to part – Zooms to the extents of the selected part. This is convenient if you select multiple parts.

SC Structure > Plate > Corner Treatment > Remove

Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Structure Corner Treatment drop-down > Remove Corner Treatment
Menu ..... SC Structure > Plate > Corner Treatment > Remove
Toolbar..... Structure > Structure Corner Treatment Flyout >
Command ..... SCREMOVECORNERTREATMENT
Permissions ... Structure > Modeling
Procedure..... Corner Treatments (page 81)

Removes a corner treatment from a plate part. The closest corner treatment to the selected point is removed.

SC Structure > Plate > Corner Treatment > Display

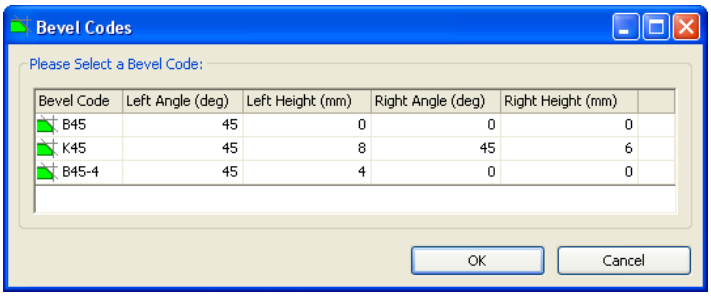
Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Structure Corner Treatment drop-down > Display Corner Treatments
Menu ..... SC Structure > Plate > Corner Treatment > Display
Toolbar..... Structure > Structure Corner Treatment Flyout >
Command ..... SCDISPLAYCORNERTREATMENT
Permissions ... Structure > Modeling
Procedure..... Corner Treatments (page 81)

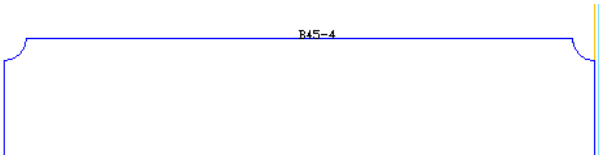
Adds display markings to the drawing to make corner treatments more visible.

SC Structure > Plate > Bevel > Create

Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Bevel drop-down > Create Bevel
Menu ..... SC Structure > Plate > Bevel > Create
Toolbar..... None
Command ..... SCBEVEL
Permissions ... Structure > Modeling
Procedures ..... Bevel Information (page 166)

Adds a bevel code to the edge of a part.





SC Structure > Plate > Bevel > Edit

Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Bevel drop-down > Edit Bevel
Menu ..... SC Structure > Plate > Bevel > Edit
Toolbar..... None
Command ..... SCEDITBEVEL
Permissions ... Structure > Modeling
Procedure..... None

Edits an existing bevel on an edge of a plate. This only allows you to change the type of bevel and not the start or end of the bevel.

SC Structure > Plate > Bevel > Remove

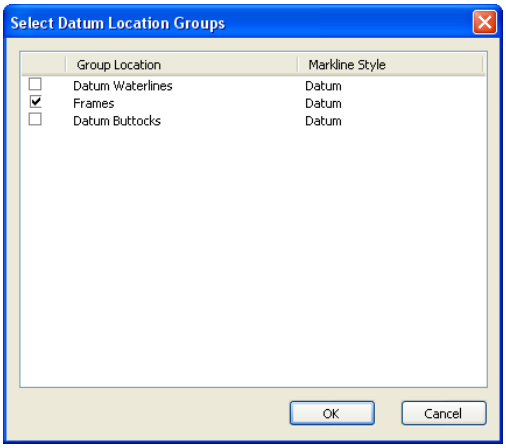
Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Bevel drop-down > Remove Bevel
Menu ..... SC Structure > Plate > Bevel > Remove
Toolbar..... None
Command ..... SCREMOVEBEVEL
Permissions ... Structure > Modeling
Procedure..... None

Removes a bevel from a plate part.

SC Structure > Plate > Manage Datum Lines

Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Manage Datum Lines
Menu ..... SC Structure > Plate > Manage Datum Lines
Toolbar..... Structure >
Command ..... SCDATUM
Permissions ... Structure > Modeling
Procedure..... None

Inserts datum lines onto or removes datum lines from plate parts. Datum lines are lines marked for locating and aligning structure during assembly.



Datum line locations and location groups are created or edited in the Hull module or from the Planar Group Drawing Properties.

SC Structure > Plate > Manage Weld Shrinkage Icon

Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Manage Weld Shrinkage Icon
Menu ..... SC Structure > Plate > Manage Weld Shrinkage Icon
Toolbar..... Structure >
Command ..... SCWELDICON
Permissions ... Structure > Modeling
Procedure..... Weld Shrinkage Icon (page 165)

Lets you insert a weld shrinkage icon.

Weld Shrinkage Icon Window

Values as

Percentage – The X Shrinkage and Y Shrinkage values are taken as a percentage of the length in the X and Y direction.

Absolute – The X Shrinkage and Y Shrinkage values are taken as an absolute value in the AutoCAD units in the X and Y direction.

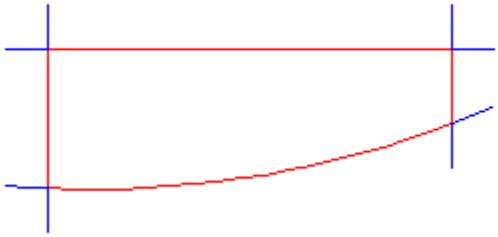
The X and Y direction are determined by the axes of the icon itself.

Icon Size – Size of the weld shrinkage icon in the drawing.

SC Structure > Plate > Invalid Boundary Diagnostic

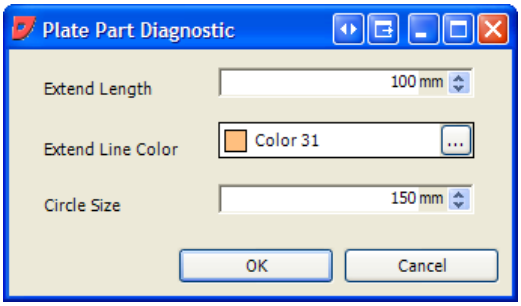
Button .....
Ribbon ..... Structure Modeling tab > Plates panel > Invalid Boundary Diagnostic
Menu ..... SC Structure > Plate > Invalid Boundary Diagnostic
Toolbar..... None
Command ..... SCPLATEPARTDIAG
Permissions ... Structure > Modeling
Procedure..... None

Helps find gaps in the boundary in invalid parts. Lines are created as extensions to every polyline.



Blue lines are added to bridge gaps. Only used ones are left behind

Settings are shown that allow you to specify how the command behaves.

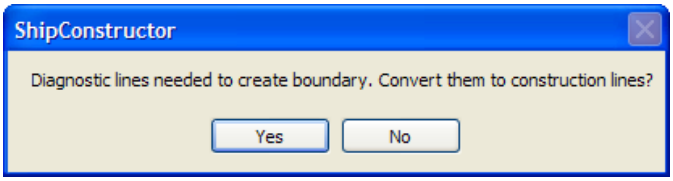


Extend Length – Length of extending lines to create

Extend Line Color – Color of the SConToolpathDiagnosticLine object left around if it would become part of the toolpath.

Circle Size – Size of the circle in the SConToolpathDiagnosticLine object. Makes it easier to locate the object visually.

If the new lines make the part valid then they are left around as custom objects or you can convert them to construction lines that can be added to the part.



If you do not convert them then SConToolpathDiagnosticLine objects are created. You can later explode them and add them to the part.

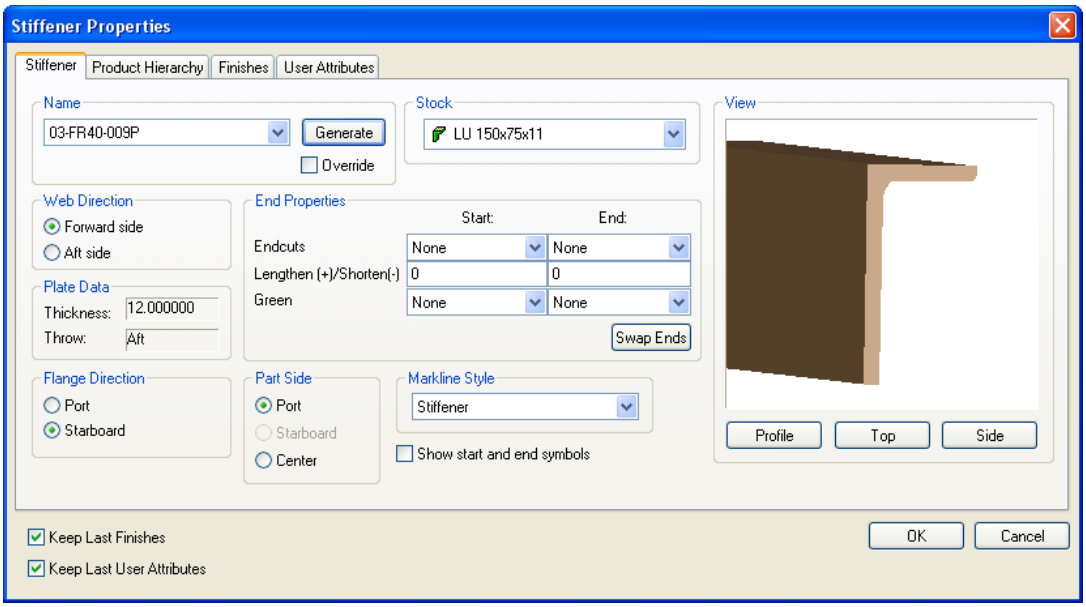
This command does not change any existing construction line or part.

SC Structure > Stiffener > New

Button .....
Ribbon ..... Structure Modeling tab > Stiffeners panel > Structure Stiffener drop-down > New Stiffener
Menu ..... SC Structure > Stiffener > New
Toolbar..... Structure > Structure Stiffener Flyout >
Command ..... SCSTIFF
Permissions ... Structure > Modeling
Procedure..... Stiffeners (page 167)

Opens the Stiffener Properties window.

Stiffener Properties Window



Name – The name that the new stiffener will use.

Generate – Generates a new name for this part using the current naming convention.

Override – Lets you manually create a name for this part, overriding the current naming convention.

Stock – The stock that will be used to create the new stiffener.

Web Direction – The direction of the web at the start of the stiffener. If you are attaching the stiffener to a plate, you will be able to choose the side of the plate that the stiffener will reside on.

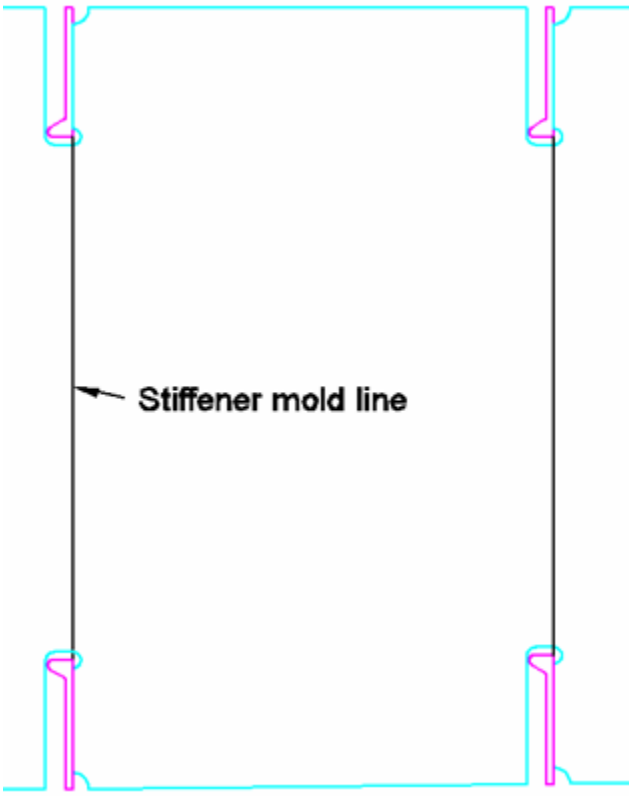
Flange Direction – The direction of the flange at the start of the stiffener.

Thickness – The thickness of the plate part you are attaching the stiffeners to. This is only visible when you are attaching the stiffeners to a plate part.

Throw – The throws of the plate part you are attaching the stiffeners to. This is only visible when you are attaching the stiffeners to a plate part.

Endcuts – The endcut that will be applied to the specified end of the stiffener.

Lengthen(+)/Shorten(-) - Allows you to lengthen or shorten the stiffener without having to modify the original stiffener moldline. Entering a positive value will lengthen the specified end by the entered amount and a negative value will shorten the specified end by the entered amount. Consider the following image. The cross sections of the longitudinal profiles have been connected with straight lines. The flatbars to be extruded along these lines have to overlap the longitudinal profiles by 50 mm. Instead of extending the lines manually, you can set the lengthen option to 50 for Start and End.



In other cases you might stay away from the end of the extrusion line. Just insert a negative value to create a stiffener that is shorter by the given amount.

Note: If only a portion of the original extrude curve was used to create the stiffener, the stiffener will be lengthened along the original extrude curve. Once the end of the original extrude curve is reached, the stiffener will be extended in the direction of the specific end of the original extrude curve.

Green – The green standard that specifies the amount of stock to be added to the stiffener when nested. Green is applied after all trims and endcuts are applied to the specific end of the stiffener. To define green standards see Structure > Profiles > Green Standards (page 361).

Swap Ends – Swaps start and end values currently for Endcuts, Lengthen(+)/Shorten(-), and Green.

Part Side – The side of the ship that the part will report that it is on.

Markline Style – The markline style that will be used for the stiffener markline when attaching the new stiffeners to plate parts. The markline style is set in Manager. See Markline Styles (page 73).

Show Start and End Symbols – Specifies whether the S and E symbols will be shown at the start and ends of all stiffeners and faceplates in the current drawing.

View – A preview of the stiffener stock and endcuts that will be used to create the new stiffener. The buttons below the preview set pre-set views for the Profile, Top and Side of the preview stiffener. Right-clicking on the Preview will bring up a menu allowing the selection of Profile, Start, End and Zoom to Extents views.

Product Hierarchy Tab – See SC Structure > Plate > New.

Finishes Tab – See SC Structure > Plate > New.

Keep Last Finishes – See SC Structure > Plate > New.

Keep Last User Attributes – See SC Structure > Plate > New.

SC Structure > Stiffener > New from Points

Button .....
Ribbon ..... Structure Modeling tab > Stiffeners panel > Structure Stiffener drop-down > New from Points
Menu ..... SC Structure > Stiffener > New from Points...
Toolbar..... Structure > Structure Stiffener Flyout >
Command ..... SCSTIFFPOINTS
Permissions ... Structure > Modeling
Procedure..... Stiffeners (page 167)

Creates a new straight stiffener without having to draw a line first.

SC Structure > Stiffener > Edit

Button .....
Ribbon ..... Structure Modeling tab > Stiffeners panel > Edit Stiffener
Menu ..... SC Structure > Stiffener > Edit
Toolbar..... Structure > Structure Stiffener Flyout >
Command ..... SCEDITSTIFF
Permissions ... Structure > Modeling
Procedure..... Stiffeners (page 167)

This function lets you change the properties of one or more stiffeners. The window is the same as for creating new stiffeners.

SC Structure > Stiffener > Attach to Plate

Button .....
Ribbon ..... Structure Modeling tab > Stiffeners panel > Attach Stiffener to Plate
Menu ..... SC Structure > Stiffener > Attach to Plate
Toolbar..... None
Command ..... SCATTACHSTIFF
Permissions ... Structure > Modeling
Procedure..... None

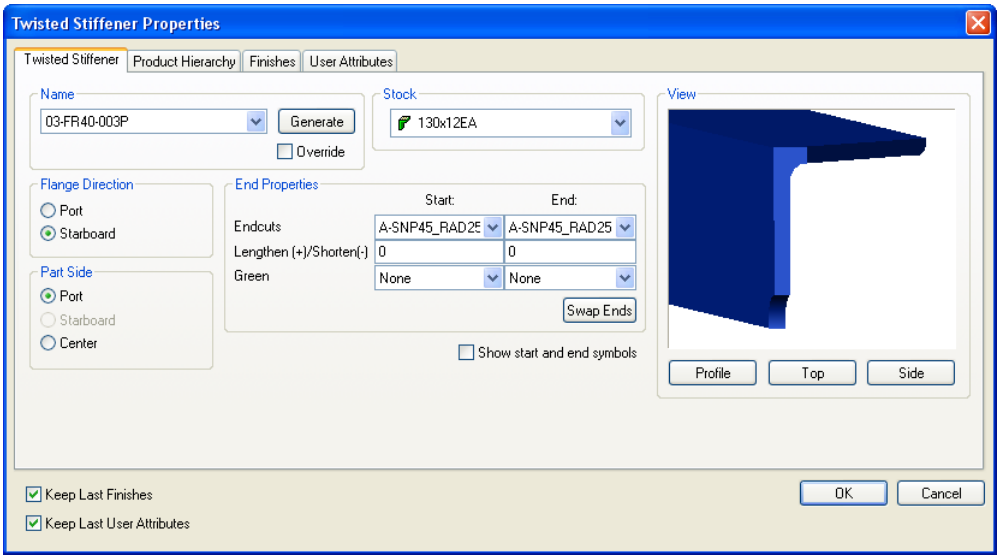
This function lets you attach a stiffener to a plate. This will create a new markline on the plate.

SC Structure > Stiffener > New Twisted

Button .....
Ribbon ..... Structure Modeling tab > Stiffeners panel > New Twisted
Menu ..... SC Structure > Stiffener > New Twisted
Toolbar..... None
Command ..... SCTWIST
Permissions ... Structure > Modeling
Procedure..... Twisted Stiffeners (page 175)

This function lets you insert a new twisted stiffener into the current group drawing. You can add the twisted stiffeners to an existing frame, deck, or girder drawing, but we recommend that you create a new drawing specifically for the twisted stiffeners. The most convenient way is to create a fake Curved Model drawing.

Twisted Stiffener Properties Window



Name – The name that the new twisted stiffener will use.

Generate – Generates a new name for this part using the current naming convention.

Override – Lets you manually create a name for this part, overriding the current naming convention.

Stock – The stock that will be used to create the twisted stiffener.

Flange Direction – The direction of the flange at the start of the twisted stiffener.

Endcuts – The endcut that will be applied to the specified end of the twisted stiffener.

Lengthen(+)/Shorten(-) – Allows you to lengthen or shorten the twisted stiffener without having to modify the original geometry. Entering a positive value will lengthen the specified end by the entered amount and a negative value will shorten the specified end by the entered amount.

Green – The green standard that specifies the amount of stock to be added to the twisted stiffener when nested. Green is applied after all trims and endcuts are applied to the specific end of the twisted stiffener. The green standards are defined in Manager. See Plate Green Standards (page 83).

Swap Ends – Swaps start and end values currently for Endcuts, Lengthen\Shorten, and Green.

Part Side – The side of the ship that the part will report that it is on

Show Start and End Symbols – Specifies whether the S and E symbols will be shown at the start and ends of all stiffeners and faceplates in the current drawing.

View – A preview of the stiffener stock and endcuts that will be used to create the new stiffener. The buttons below the preview set pre-set views for the Profile, Top, and Side of the preview stiffener. Right-clicking on the Preview brings up a menu where you select Profile, Start, End, and Zoom to Extents views.

Product Hierarchy Tab – See Plate Part Creation (page 158).

Finishes Tab – See Plate Part Creation (page 158).

Keep Last Finishes – See Plate Part Creation (page 158).

Keep Last User Attributes – See Plate Part Creation (page 158).

SC Structure > Stiffener > Edit Twisted

Button .....
Ribbon ..... Structure Modeling tab > Stiffeners panel > Edit Twisted
Menu ..... SC Structure > Stiffener > Edit Twisted
Toolbar..... None
Command ..... SCEDITTWIST
Permissions ... Structure > Modeling
Procedure..... Twisted Stiffeners (page 175)

Edits the properties of a twisted stiffener. See SC Structure > Stiffener > New Twisted (page 415) for details on the properties of the twisted stiffener.

SC Structure > Stiffener > Clean Geometry of Twisted

Button .....
Ribbon ..... Structure Modeling tab > Stiffeners panel > Clean Twisted Geometry
Menu ..... SC Structure > Stiffener > Remove Vertices Below Tolerance
Toolbar..... None
Command ..... SCCLEANTWISTEDSTIFFENER
Permissions ... Structure > Modeling
Procedure..... Twisted Stiffeners (page 175)

The purpose of this operation is to allow users to correct issues that can arise from the complexity of the twisted stiffener geometry. This operation may remove points from the parts underlying geometry so users should only use it if they need to. Under normal circumstances there should be no need to use this operation. Possible uses for this operation are:

- To reduce the density of points in the twisted stiffener model to improve performance and reduce complexity.
- To eliminate abnormalities in the geometry which are causing the part to model or plate nest incorrectly.

SC Structure > Stiffener > Cutout > Insert

Button .....
Ribbon ..... Structure Modeling tab > Stiffeners panel > Structure Stiffener drop-down > Insert Profile Cutout
Menu ..... SC Structure > Stiffener > Cutout > Insert
Toolbar..... Structure > Structure Stiffener Flyout >
Command ..... SCEXTRADDCUTOUT
Permissions ... Structure > Modeling
Procedure..... Stiffeners (page 167)

Lets you add a cutout to a stiffener.

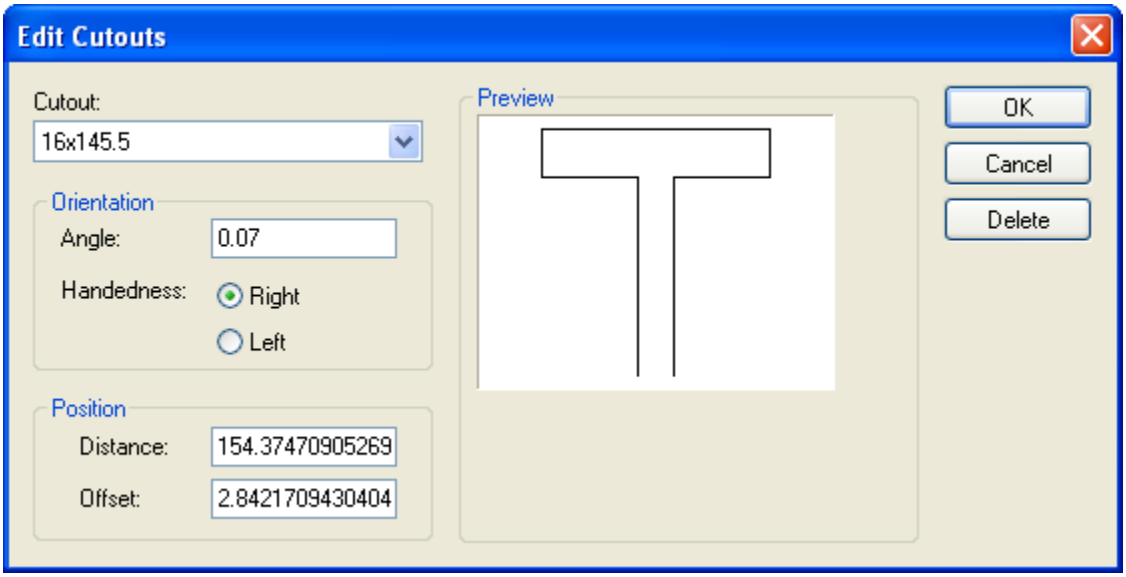
SC Structure > Stiffener > Cutout > Edit

Button .....
Ribbon ..... Structure Modeling tab > Stiffeners panel > Structure Stiffener drop-down > Edit Profile Cutout
Menu ..... SC Structure > Stiffener > Cutout > Edit

Toolbar..... Structure > Structure Stiffener Flyout >
Command ..... SCEXTREDITCUTOUT
Permissions ... Structure > Modeling
Procedure..... Stiffeners (page 167)

Lets you edit cutouts that exist on stiffeners using the Edit Cutouts window.

Edit Cutouts Window



Cutout – The name of the cutout profile being used to make this cutout.

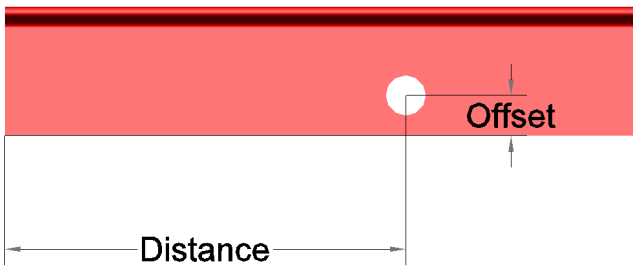
Angle – Angle in degrees from parallel to the stiffener.

Right and Left -- The handedness of the cutout. This setting allows you to mirror the cutout about the Y axis in the cutout drawing.

Note: The preview always shows the right-handed view.

Distance – The distance from the start of the stiffener.

Offset – The distance from the mold line of the stiffener along the web direction.



Delete – Removes this cutout from the stiffener.

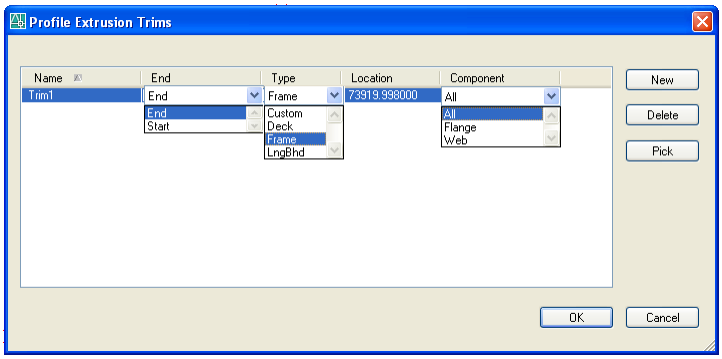
SC Structure > Stiffener > Trim

Button .....
Ribbon ..... Structure Modeling tab > Stiffeners panel > Trim Profile
Menu ..... SC Structure > Stiffener > Trim

Toolbar..... Structure > Structure Stiffener Flyout >
Command ..... SCTRIMEXTR
Permissions ... Structure > Modeling
Procedure..... Stiffeners (page 167)

A trim on a stiffener is a planar cut that slices off a piece of the stiffener. When trimming a stiffener, planes are defined by picking an orthogonal plane or a three point plane. The plane is not related to any other object, so moving the stiffener after will not recalculate the trim. The trim will maintain its relative position to the stiffener. Trim angles are calculated and stored in the database for each trim applied. Please see our online knowledgebase for details on trim angles.

Stiffener Trim Window



Name – The name to identify the trim by. If there is more than one trim applied to the same end of a part, the names for each of those trims have to be unique.

End – The end of the stiffener to apply the trim to.

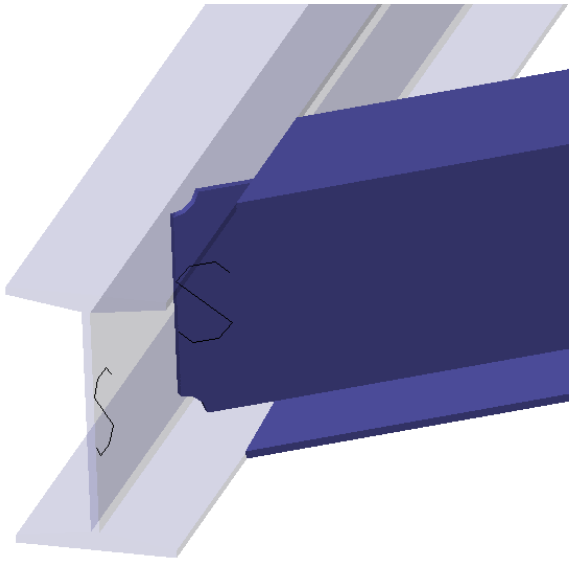
Type – The type of plane being used as the trim plane. The following options are available:

- Frame – YZ plane
- Deck – XY plane
- LngBhd – XZ plane
- Custom – Arbitrary Plane

Location – The position of the trim plane in the world coordinate system. This field is not used for trims with the type set to Custom and will be set to display N/A. The Location can either be entered manually or the Pick button can be used to select the trim plane in the drawing.

Component – The component of the stiffener (for example, flange or web). Only components that are valid for the specific stock of the part being trimmed will be shown in this list.

- All – Both the flange and the web will be trimmed.
- Flange – The trim will be applied to the entire flange and not the web.
- Top Flange – The trim will only be applied to the top flange.
- Bottom Flange – The trim will only be applied to the bottom flange.
- Web – The trim will only be applied to the web.



Stiffener-to-stiffener connection showing component trims

New – This button adds a new trim to the list. The position of the trim needs to be set before this trim is valid and can be saved.

Delete – This button will delete the trim that is currently selected in the list.

Pick – This button brings you back to the current drawing so you can select the position of the currently selected trim in the list. For trims with a type of Custom, three points are needed to define the desired trim plane. Pressing Enter instead of picking the third point will automatically use a point along the Z axis of the current UCS for the third point describing the trim plane. For trims with any other type of trim, only a single point is needed to define the trim plane.

SC Structure > Stiffener > Insert Weld Seam Reliefs

Button .....
Ribbon ..... Structure Modeling tab > Stiffeners panel > Trim Profile
Menu ..... SC Structure > Stiffener > Insert Weld Seam Reliefs
Toolbar..... None
Command ..... SCINSERTWELDSEAMRELIEFS
Permissions ... Structure > Modeling
Procedure..... Stiffeners (page 167)

Inserts cutouts into stiffeners that would be for weld seam reliefs along a straight edge.

SC Structure > Stiffener > Extract Neutral Axis

Button .....
Ribbon ..... Structure Modeling tab > Stiffeners panel > Extract Neutral Axis
Menu ..... SC Structure > Stiffener > Extract Neutral Axis
Toolbar..... None
Command ..... SCEXTRNAXIS
Permissions ... Structure > Modeling
Procedure..... Twisted Stiffeners (page 175)

Creates a new polyline from the neutral axis of the stiffener. The neutral axis is created from a polyline following the centroid of the cross sections.

SC Structure > Stiffener > Extract Mold Line

Button .....
Ribbon ..... Structure Modeling tab > Stiffeners panel > Extract Mold Line
Menu ..... SC Structure > Stiffener > Extract Mold Line
Toolbar..... None
Command ..... SCEXTRMOLDLINE
Permissions ... Structure > Modeling
Procedure..... Twisted Stiffeners (page 175)

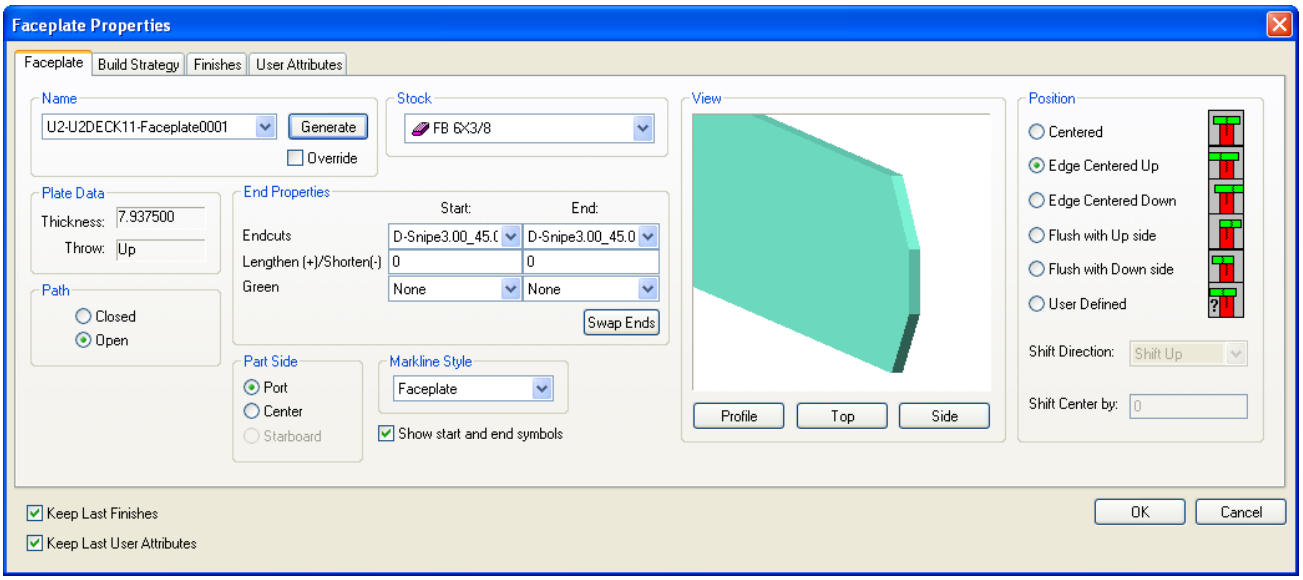
Extracts the moldline polyline used to create the stiffener.

SC Structure > Faceplate > New

Button .....
Ribbon ..... Structure Modeling tab > Faceplate panel > New Faceplate
Menu ..... SC Structure > Faceplate > New
Toolbar..... Structure > Structure Faceplate Flyout >
Command ..... SCFACEPLATE
Permissions ... Structure > Modeling
Procedure..... Faceplates (page 178)

Creates a new faceplate part.

Faceplate Properties Window



Name – The name that the new faceplate will use.

Generate – Generates a new name for this part using the current naming convention.

Override – Lets you manually create a name for this part, overriding the current naming convention.

Stock – The flatbar stock that the new faceplate will be created from.

Plate Data – Displays the Thickness and Throw of one of the plate parts the faceplate is going to be attached to.

Path – Specifies whether the faceplate will be closed or open. If Closed is selected, the faceplate will run along the complete closed loop of the selected faceplate path. If Open is selected, the faceplate will only use a portion of the selected faceplate path.

Endcuts – The endcut that will be applied to the specified end of the faceplate.

Lengthen(+)/Shorten(-) – Allows you to lengthen or shorten the faceplate without having to modify the original extrusion curve. Entering a positive value will lengthen the specified end by the entered amount and a negative value will shorten the specified end by the entered amount.

Note: If only a portion of the original extrude curve was used to create the faceplate, the faceplate will be lengthened along the original extrude curve. Once the end of the original extrude curve is reached the faceplate will be extended in the direction of the specified end of the original extrude curve.

Green – The amount of stock to be added to the faceplate when nested. Green is applied after all trims and endcuts are applied to the specified end of the faceplate.

Swap Ends – Swaps start and end values currently for Endcuts, Lengthen\Shorten, and Green.

Markline Style – The markline style that will be used for the faceplate markline when attaching the new faceplate to plate parts.

Part Side – The side of the ship that the part will report that it is on

Show Start and End Symbols – specifies whether the S and E symbols will be shown at the start and ends of all stiffeners and faceplates in the current drawing.

Preview – A preview of the faceplate stock and endcuts that will be used to create the new faceplate. The buttons below the preview set pre-set views for the Profile, Top, and Side of the preview faceplate. Right clicking on the Preview will bring up a menu allowing the selection of Profile, Start, End and Zoom to Extents views.

Position – Specifies the position of the faceplate relative to the position of the plate part it is attached to.

- Centered – The center of the faceplate will be aligned with the center of the plate part that it is attached to.
- Edge Centered – The center of the faceplate will be aligned with the specified edge of the plate part.
- Flush with Side – The edge of the faceplate will be aligned with the specified edge of the plate part. The edge of both the faceplate and the plate part will always match. For example, if Flush with Up Side is chosen, the top side of the faceplate is aligned to be flush with the top side of the plate part.
- User Defined – Allows the user to define the alignment of the faceplate manually.
- Shift Direction – Specifies the direction of the user-defined alignment. This is only used when the User Defined option is selected.
- Shift Center by – Specifies the offset amount of the user defined alignment. The faceplate's position will be shifted by the amount specified in this field in the direction selected in the Shift Direction field. This is only used when the User Defined option is selected.

Product Hierarchy Tab – See Plate Part Creation (page 158).

Finishes Tab – See Plate Part Creation (page 158).

Keep Last Finishes – See Plate Part Creation (page 158).

Keep Last User Attributes – See Plate Part Creation (page 158).

SC Structure > Faceplate > Edit

Button .....
Ribbon ..... Structure Modeling tab > Faceplate panel > Edit
Menu ..... SC Structure > Faceplate > Edit
Toolbar..... Structure > Structure Faceplate Flyout >
Command ..... SCFACEPLATEEDIT
Permissions ... Structure > Modeling
Procedure..... Faceplates (page 178)

This command will edit the part properties of faceplate parts.

SC Structure > Faceplate > Copy and Rotate 180

Button .....
Ribbon ..... Structure Modeling tab > Faceplate panel > Copy and Rotate 180
Menu ..... SC Structure > Faceplate > Copy and Rotate 180
Toolbar..... Structure > Structure Stiffener Flyout >
Command SCCOPYFACEPLATE180
Permissions ... Structure > Modeling
Procedure..... Faceplates (page 178)

This command creates a polar array of two. This will allow you to create two faceplates that surround a circular hole. The array command will achieve the same result.

SC Structure > Faceplate > Cutout > Insert

Button .....
Ribbon ..... Structure Modelling tab > Faceplate panel > Structure Faceplate drop-down > Insert Profile Cutout
Menu ..... SC Structure > Faceplate > Cutout > Insert
Toolbar..... Structure > Structure Faceplate Flyout >
Command ..... SCEXTRADDCUTOUT
Permissions ... Structure > Modeling
Procedure..... Faceplates (page 178)

Inserts a cutout into a faceplate. See SC Structure > Stiffener > Cutout > Insert (page 416).

SC Structure > Faceplate > Cutout > Edit

Button .....
Ribbon ..... Structure Modelling tab > Faceplate panel > Structure Faceplate drop-down > Edit Profile Cutout
Menu ..... SC Structure > Faceplate > Cutout > Edit
Toolbar..... Structure > Structure Faceplate Flyout >
Command ..... SCEXTREDITCUTOUT
Permissions ... Structure > Modeling
Procedure..... Faceplates (page 178)

Allows you to edit the cutouts in a faceplate. See SC Structure > Stiffener > Cutout > Edit (page 417).

SC Structure > Faceplate > Trim

Button .....
Ribbon ..... Structure Modelling tab > Faceplate panel > Trim Profile
Menu ..... SC Structure > Faceplate > Trim
Toolbar..... Structure > Structure Faceplate Flyout >
Command ..... SCTRIMEXTR
Permissions ... Structure > Modeling
Procedure..... Faceplates (page 178)

Trims an end of a faceplate. See SC Structure > Stiffener > Trim (page 418).

SC Structure > Faceplate > Extract Neutral Axis

Button .....
Ribbon ..... Structure Modeling tab > Faceplate panel > Extract Neutral Axis
Menu ..... SC Structure > Faceplate > Extract Neutral Axis
Toolbar..... None
Command ..... SCEXTRNAXIS
Permissions ... Structure > Modeling
Procedure..... Faceplates (page 178)

Extracts the neutral axis polyline from the faceplate. During normal use this command is not necessary.

SC Structure > Faceplate > Extract Mold Line

Button .....
Ribbon ..... Structure Modeling tab > Faceplate panel > Extract Mold Line
Menu ..... SC Structure > Faceplate > Extract Mold Line
Toolbar..... None
Command ..... SCEXTRMOLDLINE
Permissions ... Structure > Modeling
Procedure..... Faceplates (page 178)

Extracts the moldline of a faceplate as a polyline. During normal use this command is not necessary.

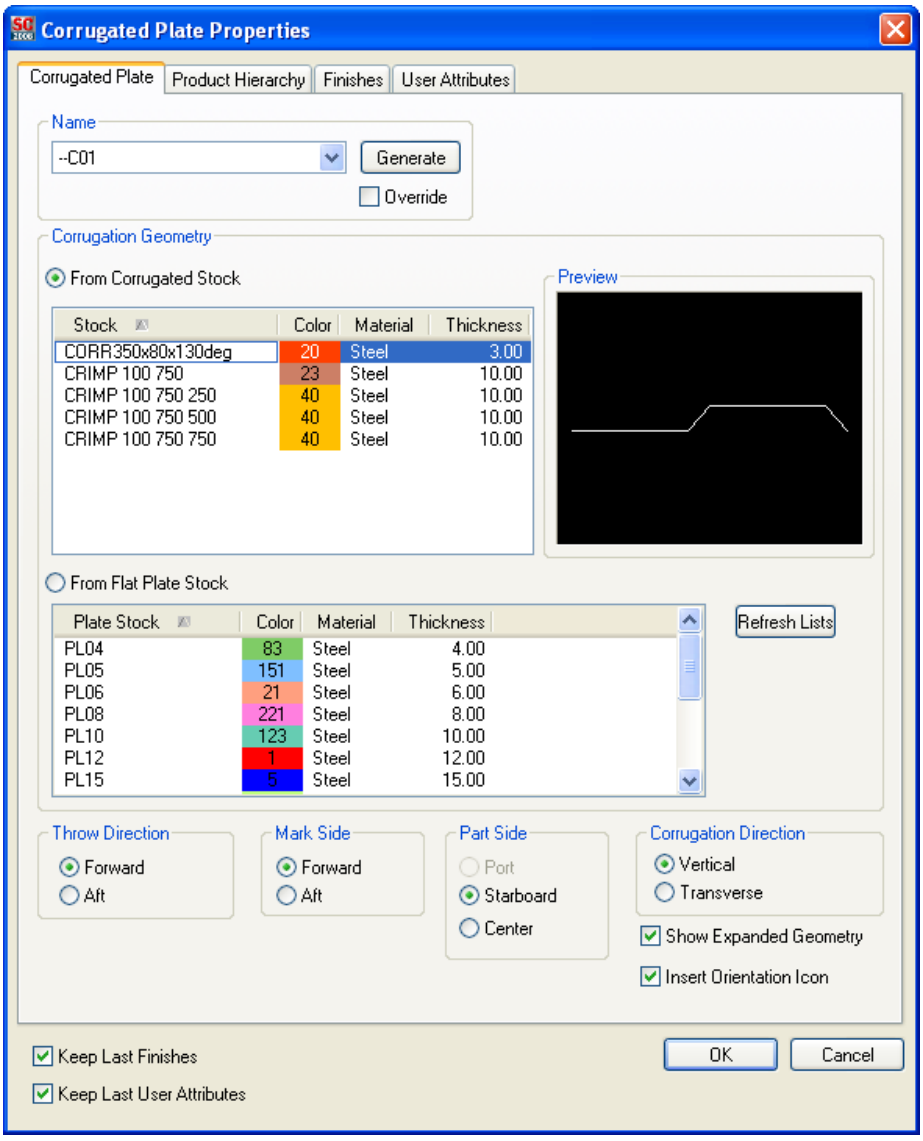
SC Structure > Corrugated Plate > New

Button .....
Ribbon ..... Structure Modeling tab > Corrugated Plate panel > New Corrugated
Menu ..... SC Structure > Corrugated Plate > New
Toolbar..... Structure > Structure Plate Flyout >
Command ..... SCCORRUGATE
Permissions ... Structure > Modeling
Procedure..... Corrugated Plates (page 180)

Creates a new corrugated plate part.

The production toolpaths are simplified with arcs when possible using the Production Tolerance. This uses the same algorithm to convert line segments to arcs as the Remove Vertices Below Tolerance command.

Corrugated Plate Properties Window



Name – The name that the new corrugated plate will use.

Generate – Generates a new name for this part using the current naming convention.

Override – Lets you manually create a name for this part, overriding the current naming convention.

From Corrugated Stock – Enables you to select the pre-defined corrugated plate stocks from the list provided to use in the creation of the corrugated plate part. The pre-defined corrugated plate stocks can be defined using Manager. The Preview displays the profile of the currently selected corrugated plate stock when this option is enabled.

From Flat Plate Stock – Enables you to select the plate stock from the list to use to create the corrugated plate part. You will need to have a polyline outlining the entire length of the corrugated profile you want to use for the corrugated plate part already present in your drawing. The corrugated profile polyline should be drawn and placed so that extruding it would intersect the entire outer toolpath (the solid is created from the intersection of the extrusions of the profile and the outer toolpath). The Preview will be blank when this option is enabled.

Refresh List – Gets the latest list of stocks from the database.

Throw Direction – The direction the amplitude (positive direction in the profile drawing) will face. This option is only applicable when the From Corrugated Stock option is used.

Markside – The side of the corrugated plate part that markings will be applied to.

Part Side – The side of the ship that the part will report that it is on

Corrugation Direction – The direction the plate will be corrugated.

Show Expanded Geometry – This option allows you to hide the expanded geometry in the model drawing. Expanded toolpaths including outer trim path, inner trim path, and bend lines (bend lines are straight lines used to mark the plate at the positions required to shape the corrugation from flat plate stock) are always generated when you create\edit the part. Bend line position is defined by point objects in the corrugation profile standard drawings when From Corrugated Stock option is used. When the From Flat Plate Stock option is used, bend line position can be defined by selecting points either with or after the corrugation curve is selected (you will be prompted), or by selecting the Generate Automatically option when prompted.

Insert Part Orientation Icon – Specifies whether or not a part orientation icon will be added to the corrugated plate part.

Preview – A preview of the stock selected in the corrugated stock list. This is only enabled when the From Corrugated Stock option has been selected.

Product Hierarchy Tab – See Plate Part Creation (page 158).

Finishes Tab – See Plate Part Creation (page 158).

User Attributes Tab – See Plate Part Creation (page 158).

SC Structure > Corrugated Plate > Edit

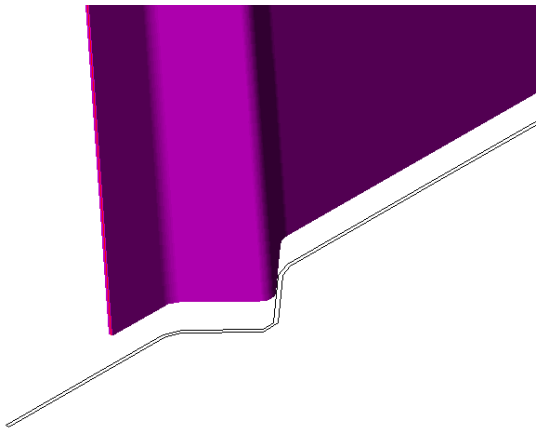
Button .....
Ribbon ..... Structure Modeling tab > Corrugated Plate panel > Edit
Menu ..... SC Structure > Corrugated Plate > Edit
Toolbar..... Structure > Structure Plate Flyout >
Command ..... SCCORRUGATEEDIT
Permissions ... Structure > Modeling
Procedure..... Corrugated Plates (page 180)

Edits the properties of an existing corrugated plate part. See SC Structure > Corrugated Plate > New (page 423) for details on the edit options.

SC Structure > Corrugated Plate > Extract Cross-section

Button .....
Ribbon ..... Structure Modeling tab > Corrugated Plate panel > Extract Cross-section
Menu ..... SC Structure > Corrugated Plate > Extract Cross-section
Toolbar..... None
Command ..... SCCORRUGATEEXTRACT
Permissions ... Structure > Modeling
Procedure..... Corrugated Plates (page 180)

Extracts the cross-section of the solid and places the resulting polyline below the corrugated plate. This is meant to be used as marking on the plate that the corrugated plate is on.



Cross-section of corrugated plate

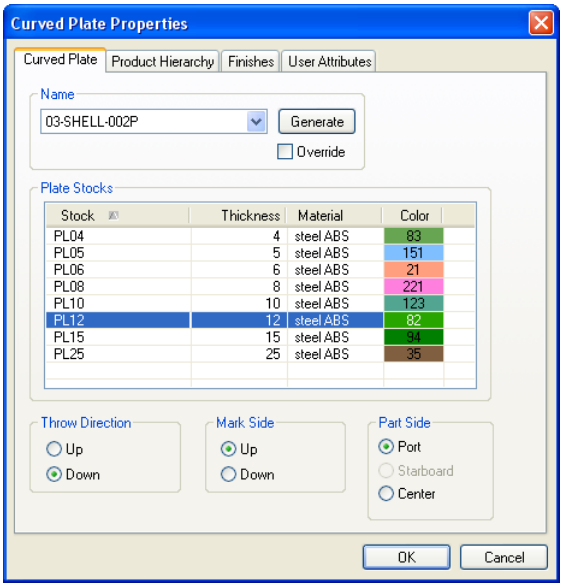
SC Structure > Curved Plate > New

Button .....
Ribbon ..... Structure Modeling tab > Curved Plate panel > New Curved
Menu ..... SC Structure > Curved Plate > New
Toolbar..... None
Command ..... SCIMPORTCURVEDPLATE
Permissions ... Structure > Modeling
Procedure..... Curved Plates (page 184)

Creates a new curved plate from an expanded plate DXF generated by ShipCAM or Maxsurf.

The production toolpaths are simplified with arcs when possible using the Production Tolerance. This uses the same algorithm to convert line segments to arcs as the Remove Vertices Below Tolerance command.

Curved Plate Properties Window



Name – The name that the new curved plate will use.

Generate – Generates a new name for this part using the current naming convention.

Override – Lets you manually create a name for this part, overriding the current naming convention.

Plate Stocks – The curved plate part will use the selected plate stock from this list.

Throw Direction – Sets the direction that the solid will throw.

Markside – The side of the curved plate part that markings will be applied to.

Part Side – The side of the ship that the part will report that it is on

Product Hierarchy Tab – See Plate Part Creation (page 158).

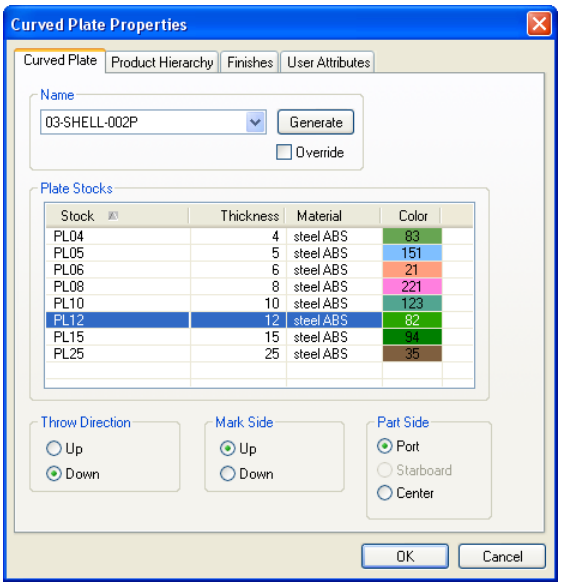
Finishes Tab – See Plate Part Creation (page 158).

User Attributes Tab – See Plate Part Creation (page 158).

SC Structure > Curved Plate > Edit

Button .....
Ribbon ..... Structure Modeling tab > Curved Plate panel > Edit
Menu ..... SC Structure > Curved Plate > Edit
Toolbar..... None
Command ..... SCEDITCURVEDPLATE
Permissions ... Structure > Modeling
Procedure..... Curved Plates (page 184)

Edits the properties of the curved plate. See SC Structure > Curved Plate > New (page 426) for details on the edit options.



Warning: Changing the stock or throw direction will not re-expand the plate if the plate is expanded about the neutral axis. To change the stock in this case would require you to change the stock to the plate in the Hull module and re-expanding the plate.

SC Structure > Curved Plate > Define New Outer Toolpath

Button .....
Ribbon ..... Structure Modeling tab > Curved Plate panel > Define New Outer Toolpath
Menu ..... SC Structure > Curved Plate > Define New Outer Toolpath
Toolbar..... None
Command ..... SCCURVEDPLATETOOLPATH
Permissions ... Structure > Modeling
Procedure..... Curved Plates (page 184)

Lets you change the polyline that is the outer toolpath for the curved plate part. It is possible that you will need to correct some unwanted effect in the expanded plate. There may be some detail that you need to add that you could not add before the plate was expanded. Curved plate boundaries are not like flat plate boundaries that consist of component construction lines. The curve plate boundary must be a single closed polyline.

Warning: Changing the outer toolpath does not change the solid or the weight and CG.

SC Structure > Curved Plate > Extract Production Info

Button .....
Ribbon ..... Structure Modeling tab > Curved Plate panel > Extract Production Info
Menu ..... SC Structure > Curved Plate > Extract Production Info
Toolbar..... None
Command ..... SCEXTRACTCURVEDPLATEPROD
Permissions ... Structure > Modeling
Procedure..... Curved Plates (page 184)

Extracts a copy of the geometry that is inside the expanded curved plate. This allows you to gain access to the geometry.

SC Structure > Curved Plate > Import Curved Plate From Rhino

Button .....
Ribbon ..... Structure Modeling tab > Curved Plate panel > Import Curved Plate From Rhino
Menu ..... SC Structure > Curved Plate > Import Curved Plate From Rhino
Toolbar..... None
Command ..... SCSTRUCTUREIMPORTRHINO
Permissions ... Structure > Modeling
Procedure..... Curved Plates (page 184)

Imports a surface from Rhino.

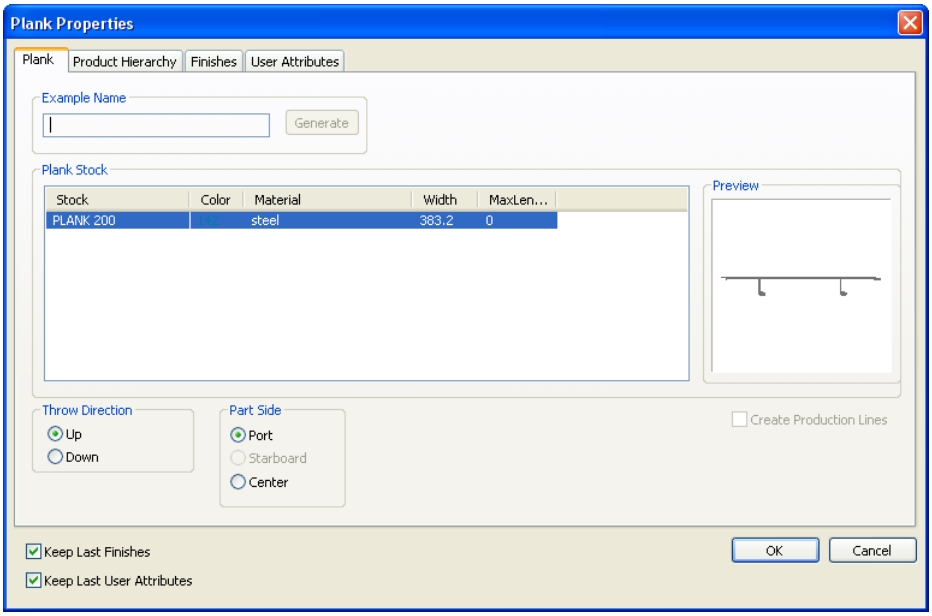
SC Structure > Planks > New

Button .....
Ribbon ..... Structure Modeling tab > Planks panel > New Plank
Menu ..... SC Structure > Planks > New
Toolbar..... Structure > Structure Plank Flyout >
Navigator..... None
Command ..... SCPLANK
Permissions ... Structure
Procedure..... Planks Creation (page 190)

Creates planks in an area with the selected planking stock.

The production toolpaths are simplified with arcs when possible using the Production Tolerance. This uses the same algorithm to convert line segments to arcs as the Remove Vertices Below Tolerance command.

Plank Properties Window



Example Name – An example of the naming to be used for the newly created plank parts. Because multiple plank parts are created and all will not be identical, multiple names are created.

Generate – Not used at this time.

Plank Stocks – The plank part will be created using the selected plank stock from this list.

Preview – A preview of the stock selected in the plank stock list.

Throw Direction - The direction the amplitude (positive direction in the profile drawing) will face.

Part Side – The side of the ship that the part will report that it is on

Create Production Lines – Specifies whether or not production information outlining the boundaries of each plank part will be generated.

Product Hierarchy Tab – See Plate Part Creation (page 158).

Finishes Tab – See Plate Part Creation (page 158).

Keep Last Finishes – See Plate Part Creation (page 158).

Keep Last User Attributes – See Plate Part Creation (page 158).

SC Structure > Planks > Edit Collection

Button .....
Ribbon ..... Structure Modeling tab > Planks panel > Edit Plank Collection
Menu ..... SC Structure > Planks > Edit Collection
Toolbar..... Structure > Structure Plank Flyout >
Navigator..... None
Command ..... SCPLANKCOLLECTIONEDIT
Permissions ... Structure
Procedure..... Planks Edit (page 191)

Edits the properties of the plank collection. A plank collection is created when you create planks. Since all the planks must maintain the same stock. Many operations are applied to the plank collection.

SC Structure > Planks > Split Plank Collection

Button .....
Ribbon ..... Structure Modeling tab > Planks panel > Split Plank Collection
Menu ..... SC Structure > Planks > Split Plank Collection
Toolbar..... Structure > Structure Plank Flyout >
Navigator..... None
Command ..... SCPLANKCOLLECTIONSPLIT
Permissions ... Structure
Procedure..... Splitting Plank Collections (page 193)

Splits the selected planks from a collection to create 2 new collections.

SC Structure > Planks > Delete Plank Collection

Button .....
Ribbon ..... Structure Modeling tab > Planks panel > Delete Plank Collection
Menu ..... SC Structure > Planks > Delete Plank Collection
Toolbar..... Structure > Structure Plank Flyout >
Navigator..... None
Command ..... SCPLANKCOLLECTIONDELETE
Permissions ... Structure
Procedure..... Deleting Plank Collections (page 193)

Deletes the selected plank collection. Links to any mirrored or identical plank collections are broken.

SC Structure > Parts > Edit

Button .....
Ribbon ..... Structure Modeling tab > Parts > Edit Part
Menu ..... SC Structure > Parts > Edit...
Toolbar..... None
Command ..... SCPARTPROP
Permissions ... Structure > Modeling
Procedure..... None

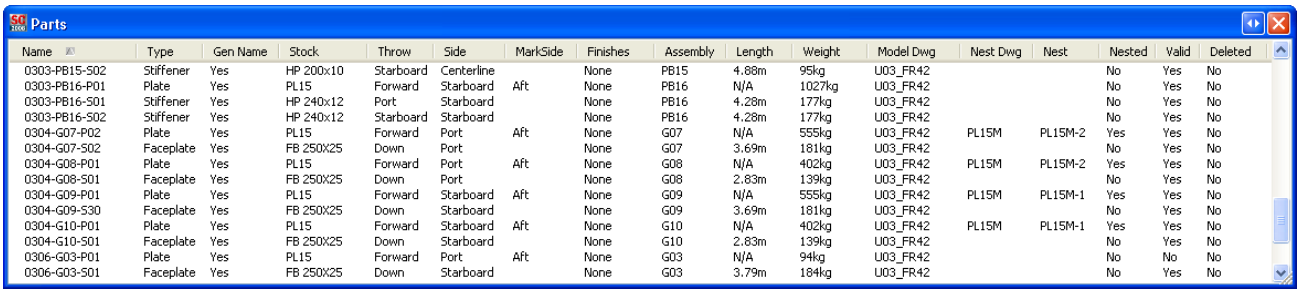
Displays the properties window for the part.

SC Structure > Parts > List

Button .....
Ribbon ..... Structure Modeling tab > Parts > List
Menu ..... SC Structure > Parts > List
Toolbar..... Structure >
Command ..... SCPARTLIST
Permissions ... Structure > Modeling
Procedure..... None

Lists the structure parts in the current drawing. This is useful for checking some of the key properties of the parts in the drawing or finding a part in the drawing. This command works in structure model drawings and nest drawings.

This window can stay visible while you work in ShipConstructor. Selecting parts in the drawing will select the part in the Parts window.



Name	Type	Gen Name	Stock	Throw	Side	MarkSide	Finishes	Assembly	Length	Weight	Model Dwg	Nest Dwg	Nest	Nested	Valid	Deleted
0303-PB15-S02	Stiffener	Yes	HP 200x10	Starboard	Centerline		None	PB15	4.88m	95kg	U03_FR42			No	Yes	No
0303-PB16-P01	Plate	Yes	PL15	Forward	Starboard	Aft	None	PB16	N/A	1027kg	U03_FR42			No	Yes	No
0303-PB16-S01	Stiffener	Yes	HP 240x12	Port	Starboard		None	PB16	4.28m	177kg	U03_FR42			No	Yes	No
0303-PB16-S02	Stiffener	Yes	HP 240x12	Starboard	Starboard		None	PB16	4.28m	177kg	U03_FR42			No	Yes	No
0304-G07-P02	Plate	Yes	PL15	Forward	Port	Aft	None	G07	N/A	555kg	U03_FR42	PL15M	PL15M-2	Yes	Yes	No
0304-G07-S02	Faceplate	Yes	FB 250X25	Down	Port		None	G07	3.69m	181kg	U03_FR42			No	Yes	No
0304-G08-P01	Plate	Yes	PL15	Forward	Port	Aft	None	G08	N/A	402kg	U03_FR42	PL15M	PL15M-2	Yes	Yes	No
0304-G08-S01	Faceplate	Yes	FB 250X25	Down	Port		None	G08	2.83m	139kg	U03_FR42			No	Yes	No
0304-G09-P01	Plate	Yes	PL15	Forward	Starboard	Aft	None	G09	N/A	555kg	U03_FR42	PL15M	PL15M-1	Yes	Yes	No
0304-G09-S30	Faceplate	Yes	FB 250X25	Down	Starboard		None	G09	3.69m	181kg	U03_FR42			No	Yes	No
0304-G10-P01	Plate	Yes	PL15	Forward	Starboard	Aft	None	G10	N/A	402kg	U03_FR42	PL15M	PL15M-1	Yes	Yes	No
0304-G10-S01	Faceplate	Yes	FB 250X25	Down	Starboard		None	G10	2.83m	139kg	U03_FR42			No	Yes	No
0306-G03-P01	Plate	Yes	PL15	Forward	Port	Aft	None	G03	N/A	94kg	U03_FR42			No	No	No
0306-G03-S01	Faceplate	Yes	FB 250X25	Down	Starboard		None	G03	3.79m	184kg	U03_FR42			No	Yes	No

Name – The name of the part.

Type – The type of part.

Gen Name – Yes means that the part name follows the naming convention. No means that the part name overrides the naming convention.

Stock – The stock of the part.

Throw – The throw direction of the part.

Side – The part side of the part.

MarkSide – The side of the plate part to mark. This is handy to verify that the marksides are correct.

Finishes – The finishes that are applied to the part.

Assembly – the short assembly name of the build strategy assembly that the part is assigned to.

Right-Click Menu

Zoom To – Zooms to the selected objects.

View in Nest – If the part is nested, this command opens the nest drawing and zooms to the part. This command applies to only plate parts and flatbar stiffeners that are plate nested.

Add Objects – See SC Structure > Plate > Add Objects (page 404) (Plate parts only).

Delete – Erases the selected parts.

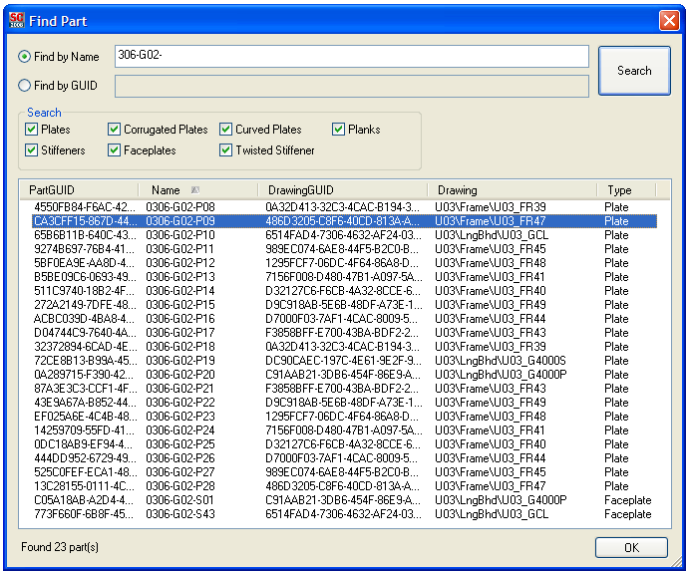
Properties – Shows the Edit Properties window for the selected part.

SC Structure > Parts > Find

Button .....
Ribbon ..... Structure Modeling tab > Parts > Find
Menu ..... SC Structure > Parts > Find
Toolbar..... None
Command ..... SCFINDPARTDWG
Permissions ... Structure > Modeling
Procedure..... None

Displays the Find Part window that allows for searching a part name for the corresponding drawing. Enter all or some of a part name, and click Search. A list of all the parts that match that name (and type) is displayed, along with the drawing name, part type, and unique database identifiers of the part and drawing. The latter two are not important for most users.

Double-clicking on a part in the results list will prompt to open the model drawing that part is in.



SC Structure > Parts > Relationships

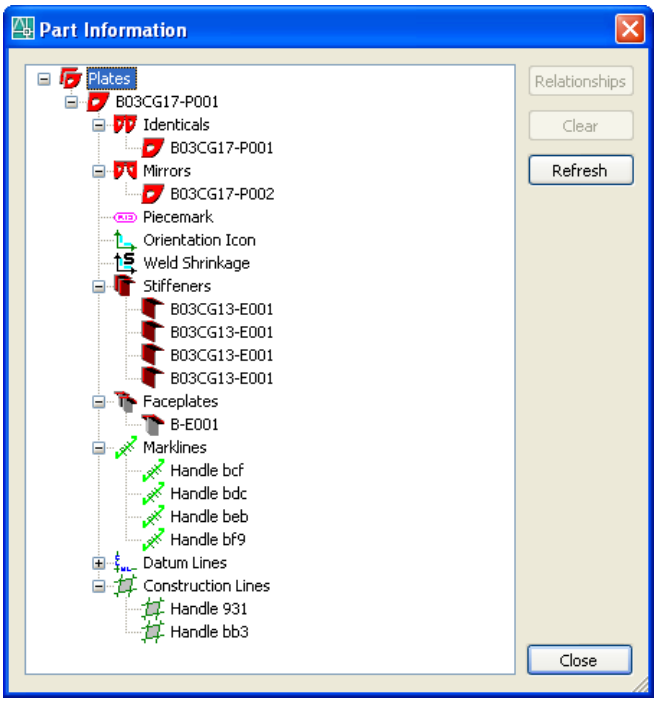
Button .....
Ribbon ..... Structure Modeling tab > Parts > Relationships
Menu ..... SC Structure > Parts > Relationships
Toolbar..... None
Command ..... SCDISPLAYRELATIONSHIPS
Permissions ... Structure > Modeling
Procedure..... None

Shows the Relationships window that shows all the related objects.

SC Structure > Parts > Information

Button .....
Ribbon ..... Structure Modeling tab > Parts > Information
Menu ..... SC Structure > Parts > Information
Toolbar..... None
Command ..... SCPARTINFO
Permissions ... Structure > Modeling
Procedure..... Removing Links Between Parts (page 156)

Shows the Part Information window that shows all the associated objects.



Relationships – Launches the Relationships window for the selected object.

Clear – Removes the selected part from the tree.

Refresh – Updates the information in the tree. This is useful to show the up-to-date information when you modify the part or its relationships in ShipConstructor.

Components of the Part Information Tree

The contents of the tree differ depending on the part type selected. In this case the following describes the case when the information about a plate part is shown.

Plate Part Name – the name of the plate part

Right Click Menu > Zoom to Part – Zooms to the extents of the selected part.

Right Click Menu > Edit Properties – Edit the properties of the selected part.

Right Click Menu > Add Objects – Runs the command SC Structure > Plate > Add Objects (page 404)

Right Click Menu > Attach Stiffener to Plate – Runs the command SC Structure > Stiffener > Attach to Plate.

Identicals – The parts that are linked as being identical parts.

Right Click Menu > Edit Properties – Edits the properties of the selected part.

Mirrors – The parts that are linked as mirror parts.

Right Click Menu > Edit Properties – Edits the properties of the selected part.

Piecemark

Right Click Menu > Edit Properties – Edits the properties of the piecemark.

Orientation Icon

Right Click Menu > Edit Properties – Edits the properties of the orientation icon.

Weld Shrinkage

Right Click Menu > Edit Properties – Edits the properties of the weld shrinkage. See SC Structure > Plate > Manage Weld Shrinkage Icon (page 410).

Flanges

Right Click Menu > Edit Properties – Edits the properties of the Flange. See SC Structure > Plate > Edit Flange (page 402).

Plates

Right Click Menu > Edit Properties – Edits the properties of the plate that is attached to the selected stiffener. See SC Structure > Plate > Edit (page 400).

Stiffeners

Right Click Menu > Edit Properties – Edits the properties of the stiffener that is attached to the selected plate. See SC Structure > Stiffener > Edit (page 414).

Faceplates

Right Click Menu > Edit Properties – Edits the properties of the faceplate that is attached to the selected plate. See SC Structure > Faceplate > Edit (page 421).

Marklines

Right Click Menu > Edit Properties – Edits the properties of the markline. See SC Structure > Parts > Edit Component (page 435).

Right Click Menu > Construction Line Info – Shows the Construction Line Info window.

Datum Lines

Right Click Menu > Edit Properties – Edits the properties of the selected datum line markline.

Construction Lines

Right Click Menu > Zoom to Construction Line – Zooms to the selected construction line

Right Click Menu > Construction Line Info – Shows the Construction Line Info window.

Right Click Menu > Remove from Plate – Removes the selected construction line from the toolpath of the plate. This could make the part invalid if no boundary can be determined.

Right Click Menu > Edit Offset – Offset construction lines only – Change the offset value of the offset construction line.

Right Click Menu > Clear Trims – Offset construction lines only – Remove all trims and extensions from the offset construction line.

Cutouts

Right Click Menu > Zoom to Plate – Zooms to the extents of the selected plate part.

Right Click Menu > Zoom to Stiffener – Zooms to the selected stiffener, which has a cutout on the current plate part

Right Click Menu > Edit Plate Properties – Opens the plate properties dialog. See SC Structure > Plate > Edit (page 400).

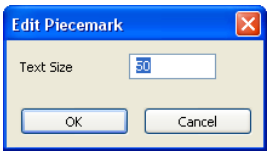
Right Click Menu > Edit Stiffener Properties – Opens the stiffener properties dialog. See SC Structure > Stiffener > Edit (page 414).

Right Click Menu > Cutout Manager – Opens the Cutout Manager dialog, See SC Structure > Plate > Manage Cutouts... (page 406)

SC Structure > Parts > Edit Component

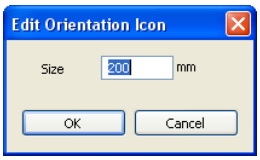
Button .....
Ribbon ..... Structure Modeling tab > Parts > Edit Component
Menu ..... SC Structure > Parts > Edit Component
Toolbar..... Structure >
Command ..... SCEDITCOMPONENT
Permissions ... Structure > Modeling
Procedure..... Add, Remove, and Edit Objects on an Existing Plate Part (page 160)

Allows you to edit components of parts.



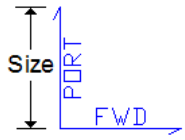
Piecemark component options

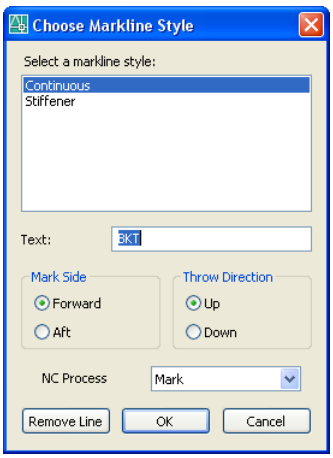
Text Size – The size of the text displayed on the piecemark.



Orientation icon component options

Size – The new size of the orientation icon. If the size is changed on one part, then changing the stardard size will not affect this part. Changing the size back to the standard size will re-enable the part to use the standard size.





Markline component options

Markline Style – The list of available styles to use. This list is made from the assigned marklines to General category.

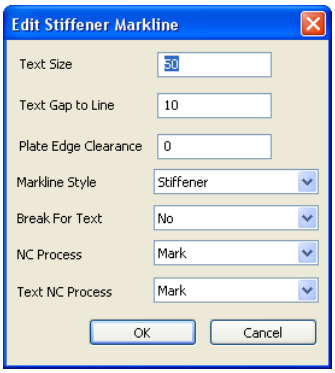
Text – Optional text string to label the markline.

Mark Side – The side of the plate that this markline will be marked.

Throw Direction – The positive side of the markline. This controls the marksymbol's orientation.

NC Process – The NC process of the markline. This overrides the markline style setting for this markline.

Remove Line – Removes the current markline from the plate. The associated construction line will remain.



Stiffener Markline component options

Text Size – The size of the text that accompanies the stiffener markline. If this value differs from the text size specified in the markline style, then both the text size and the nearside/farside indicator text size will be overridden with this value.

Text Gap to Line – The distance between the markline and its associated text.

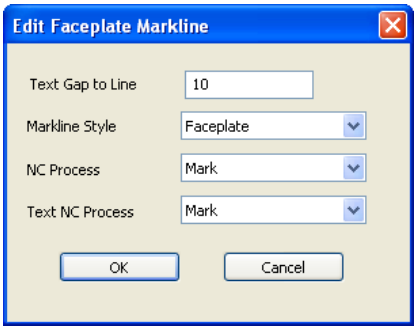
Plate Edge Clearance – The maximum allowable distance between the plate edge and the markline ends.

Markline Style – The list of available styles to use. This list is made from the assigned marklines to the Stiffener category.

Break For Text – If Yes, the line is broken apart where the line text is placed.

NC Process – The NC process of the markline when NC cutting the part.

Text NC Process – The NC process of the markline text when NC cutting the part.



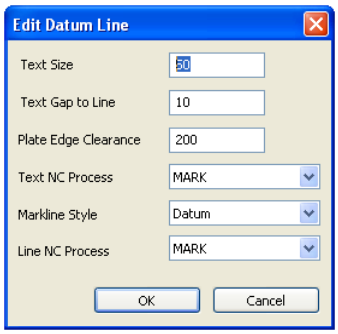
Faceplate Markline component options

Text Gap to Line – The distance between the markline and its associated text.

Markline Style – The list of available styles to use. This list is made from the assigned marklines to the Faceplate category.

NC Process – The NC process of the markline when NC cutting the part.

Text NC Process – The NC process of the markline text when NC cutting the part.



Datum Markline component options

Text Size – The size of the text that accompanies the datum line. If this value differs from the text size specified in the markline style, then both the text size and the nearside/farside indicator text size will be overridden with this value.

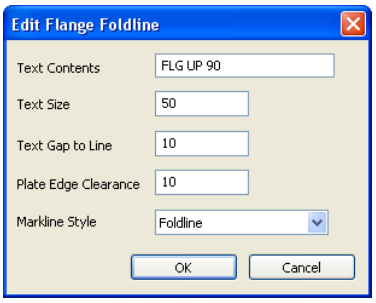
Text Gap to Line – The distance between the markline and its associated text.

Plate Edge Clearance – The maximum allowable distance between the plate edge and the markline ends.

Text NC Process – The NC process of the markline text when NC cutting the part.

Markline Style – The list of available styles to use. This list is made from the assigned marklines to the Datum category.

Line NC Process – The NC process of the markline when NC cutting the part.



Flange foldline Markline component options

Text – Optional text string to label the flange foldline.

Text Size – The size of the text that accompanies the flange foldline. If this value differs from the text size specified in the markline style, then both the text size and the nearside/farside indicator text size will be overridden with this value.

Text Gap to Line – The distance between the markline and its associated text.

Plate Edge Clearance – The maximum allowable distance between the plate edge and the markline ends.

Markline Style – The list of available styles to use. This list is made from the assigned marklines to the Flange Foldline category.

Weld Shrinkage component options

Apply Weld Shrinkage – If left unchecked, the weld shrinkage icon will not be displayed on the plate.

Values as – The format that the weld shrinkage values will take when displayed, either as percentages or as lengths using the units specified in Project Settings.

X Shrinkage – The shrinkage value displayed on the x-axis.

Y Shrinkage – The shrinkage value displayed on the y-axis.

Icon Size – The new size of the weld shrinkage icon. If the size is changed on one part, then changing the stardard size will not affect this part. Changing the size back to the standard size will re-enable the part to use the standard size.

Note: If the X Shrinkage and Y Shrinkage values are both zero, then the weld shrinkage icon will not be displayed.

SC Structure > Parts > Extract Components

Button .....
Ribbon ..... Structure Modeling tab > Parts > Extract Components
Menu ..... SC Structure > Parts > Extract Components
Toolbar..... None
Command ..... SCEXTRACTCOMPONENTS
Permissions ... Structure > Modeling
Procedure..... None

Explodes the part without deleting the original object and moves the exploded entities to a new layer.

SC Structure > Parts > Add Manual Cutouts

Button .....
Ribbon ..... Structure Modeling tab > Parts > Add Manual Cutout
Menu ..... SC Structure > Parts > Add Manual Cutouts
Toolbar..... Structure >
Command ..... SCADDMANUALCUTOUT
Permissions ... Structure > Modeling
Procedure..... None

Manually add cutouts to parts.

SC Structure > Parts > Add Dynamic Marking Block

Button .....
Ribbon ..... Structure Modeling tab > Parts > Add Dynamic Marking Block
Menu ..... SC Structure > Parts > Add Dynamic Marking Block
Toolbar..... Structure > None
Command ..... SCADDDMB
Permissions ... Structure > Modeling
Procedure..... Add a Dynamic Marking Block (page 197)

Adds a dynamic marking block (DMB) to a structure part. Currently only plates, corrugated plates, and curved plates are supported. Before a DMB can be added at least one dynamic marking block standard must be created. Standards are created through the DMB catalog in manager Structure > Dynamic Marking Blocks (page 362).

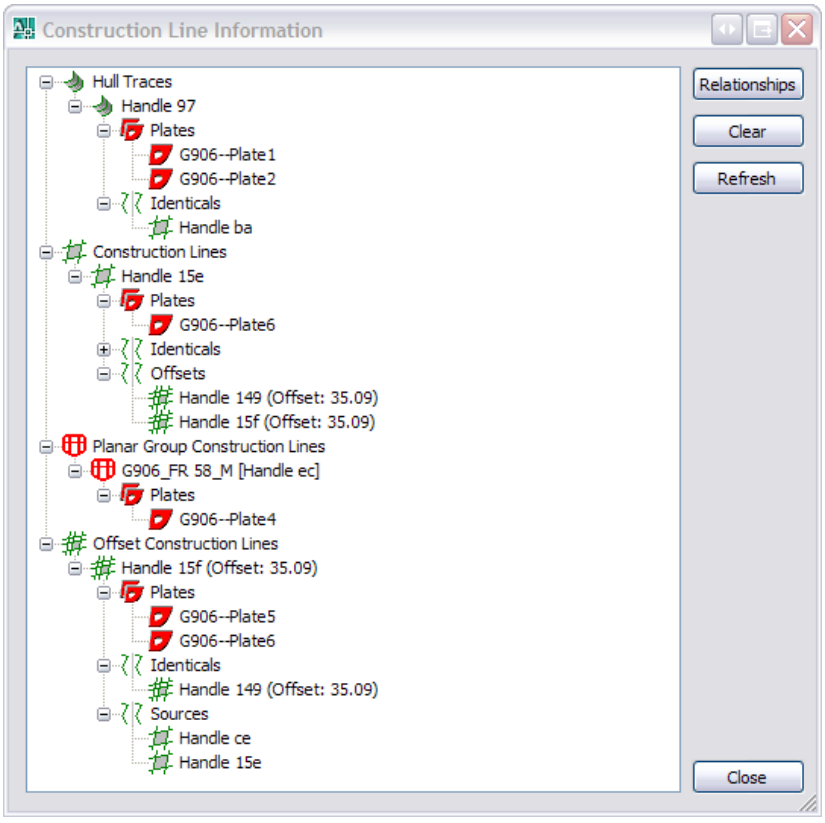
SC Structure > Parts > Boundary Diagnostic

Button .....
Ribbon ..... Structure Modeling tab > Parts > Boundary Diagnostic
Menu ..... SC Structure > Parts > Boundary Diagnostic
Toolbar..... None
Command ..... SCBOUNDARYDIAGNOSTIC
Permissions ... Structure > Modeling
Procedure..... None

SC Structure > Construction Line > Information

Button .....
Ribbon ..... Structure Modeling tab > Construction Lines > Information
Menu ..... SC Structure > Construction Line > Information
Toolbar..... None
Command ..... SCLINEINFO
Permissions ... Structure > Modeling
Procedure..... General Use Tips (page 151)
.....Construction Line Management (page 150)

Shows the information about a construction line and which construction lines and parts are related to it. The Construction Line Information window can be left open while working in ShipConstructor. Since construction lines do not have names, they are identified by their handle. This handle refers to the handle of the AutoCAD object and can be seen by listing the object.



Relationships – Opens relationship window for selected item.

Clear – Removes the selected construction line from the tree.

Refresh – Updates the information in the tree. This is useful to show the up-to-date information when you modify the construction line or its relationships in ShipConstructor.

Components of the Construction Line Information Tree

Hull Traces – the list of SConStructHullTraceConstructionLine objects.

Construction Lines – the list of SConStructUserConstructionLine objects.

Planar Group Construction Lines – the list of SConStructGroupPlaneConstructionLine objects.

Offset Construction Lines – the list of SConStructOffsetConstructionLine objects.

Plates – The list of plates that use this construction line.

Right Click Menu > Zoom to Part – Zooms to the selected plate.

Right Click Menu > Edit Properties – Edits the properties of the plate that is attached to the plate. See SC Structure > Plate > Edit (page 400)

Right Click Menu > Part Info – Shows the Part Information window.

Stiffeners - The list of stiffeners that use this construction line.

Right Click Menu > Zoom to Part – Zooms to the selected stiffener.

Right Click Menu > Edit Properties – Edits the properties of the stiffener that is attached to the plate. See SC Structure > Stiffener > Edit (page 414)

Right Click Menu > Part Info – Shows the Part Information window.

Faceplates - The list of faceplates that use this construction line.

Right Click Menu > Zoom to Part – Zooms to the selected faceplate.

Right Click Menu > Edit Properties – Edits the properties of the faceplate that is attached to the plate. See SC Structure > Faceplate > Edit (page 421)

Right Click Menu > Part Info – Shows the Part Information window.

Identicals – the list of construction lines that are identical. Mirrored construction lines are considered to be identical so there is no separate Mirrors item in the tree. Offset construction lines with the same or identical sources are also considered identical.

Offsets – lists any offset construction lines of this construction line. Displays all offset of this line and all offsets of any identical construction lines.

Sources – for an offset construction line, this lists the source construction line and any identicals to the source construction line.

SC Structure > Construction Line > Relationships

Button .....
Ribbon ..... Structure Modeling tab > Construction Lines > Relationships
Menu ..... SC Structure > Contruction Line > Relationships
Toolbar..... None
Command ..... SCDISPLAYRELATIONSHIPS
Permissions ... Structure > Modeling
Procedure..... None

Shows the Relationships window that shows all the related objects.

SC Structure > Construction Line > Convert to User Construction Line

Button .....
Ribbon ..... Structure Modeling tab > Construction Lines > Convert Object to Construction Line
Menu ..... SC Structure > Construction Line > Convert to User Construction Line
Toolbar..... None
Command ..... SCCONVERTTOCL
Permissions ... Structure > Modeling
Procedure..... None

Converts a line, circle, arc, polyline, or ellipse to a user construction line. This means that geometry will be saved in the database. Also used to convert a hull trace, planar group plane, or offset construction line to a user construction line.

SC Structure > Construction Line > Edit Offset

Button .....
Ribbon ..... Structure Modeling tab > Construction Lines > Edit Offset
Menu ..... SC Structure > Construction Line > Edit Offset
Toolbar..... None
Command ..... SCEDITOFFSET
Permissions ... Structure > Modeling
Procedure..... None

Change the offset value of an offset construction line.

SC Structure > Construction Line > Clear Trims

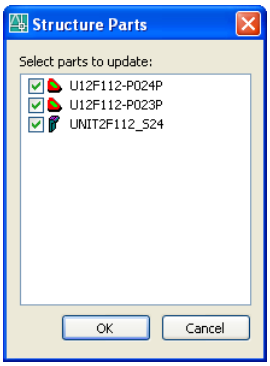
Button .....
Ribbon ..... Structure Modeling tab > Construction Lines > Clear Trims
Menu ..... SC Structure > Construction Line > Clear Trims
Toolbar..... None
Command ..... SCCLEARTRIMS
Permissions ... Structure > Modeling
Procedure..... None

Remove all trims and extensions from an offset construction line.

SC Structure > Construction Line > Swap Construction Line

Button .....
Ribbon ..... Structure Modeling tab > Construction Lines > Swap Construction Line
Menu ..... SC Structure > Construction Line > Swap Construction Line
Toolbar..... Structure >
Command ..... SCSWAPCL
Permissions ... Structure > Modeling
Procedure..... Construction Line Relationships (page 154)

Exchanges the use of one construction line in parts for another curve or construction line. It will show a list of all parts using the original construction line. All parts that are checked when the OK button is pressed will use the new construction line.



A list of parts to change

If the new curve is not a construction line then it will be converted to a user construction line. The original construction line is left unchanged.

SC Structure > Construction Line > Replace Construction Line Geometry

Button .....
Ribbon ..... Structure Modeling tab > Construction Lines > Replace Construction Line Geometry
Menu ..... SC Structure > Construction Line > Replace Construction Line Geometry
Toolbar..... Structure >
Command ..... SCREPLACECLGEOMETRY
Permissions ... Structure > Modeling
Procedure..... HullTrace Construction Line (page 153)

Replaces the geometry of a construction line with the geometry of another. The construction line retains its original identity (relationships, parts it belongs to, etc.). The other line is erased (if possible) and any parts using the original line will continue to use the line in its new position and geometry. This command is especifally useful for modifying the position of a hull trace construction line, but will run on construction lines of any type.

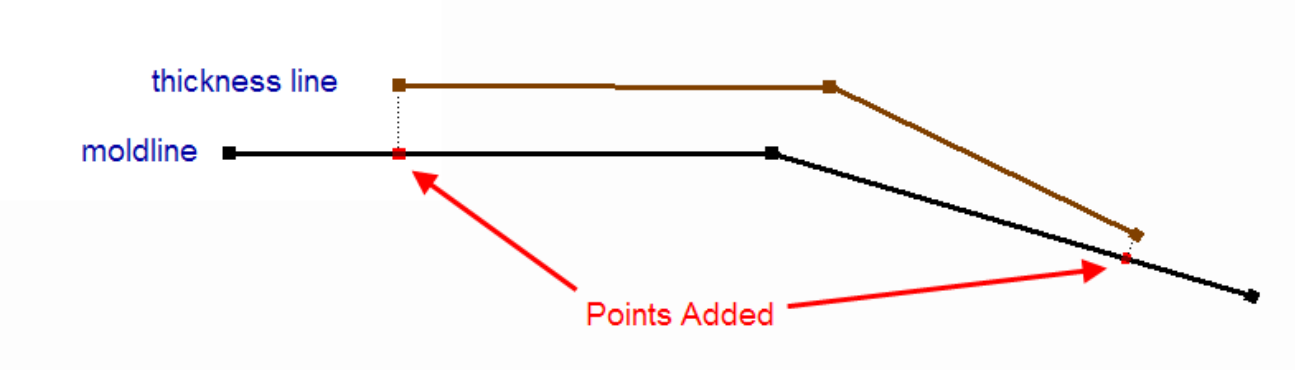
SC Structure > Construction Line > Add Bevel Angles

Button .....
Ribbon ..... Structure Modeling tab > Construction Lines > Bevel Angles drop-down > Add Bevel Angles
Menu ..... SC Structure > Construction Line > Add Bevel Angles
Toolbar..... None
Command ..... SCCLADDBEVEL
Permissions ... Structure > Modeling
Procedure..... None

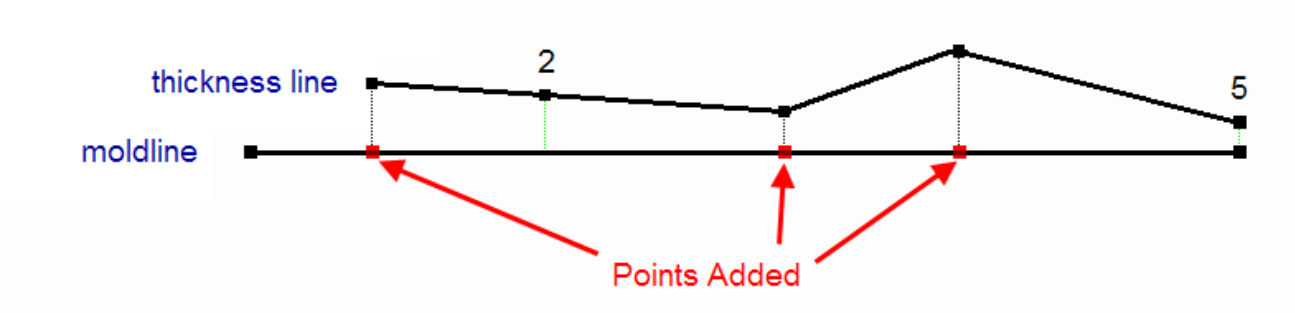
Creates a beveled construction line from two construction lines or curves. A user options exists to update or break identical construction lines. Parts that use these construction lines are updated.

These construction line / bevel angle commands are designed to be used when automatic bevels are enabled. These commands can be used to manual change bevel angle information on a constructionline when necessary. In general though, bevel angles are calculated automatically from the hull drawing.

If the thickness side line does not extend to the ends of the moldline construction line then new points are added to the moldline construction line where the bevel starts and ends.



If the thickness line contains points that have different resulting bevel angles then those points are added.



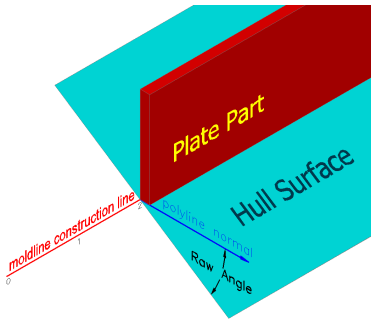
Point 2 is not added since its bevel angle is the same as if it were calculated.

An option exists to retain existing angles. This is useful when updating the angles for only a portion of a construction line. In this case angles that are for points outside the range of the thickness line are left alone.

SC Structure > Construction Line > Show Bevel Angles

Button .....
Ribbon ..... Structure Modeling tab > Construction Lines > Bevel Angles drop-down > Show Bevel Angles
Menu ..... SC Structure > Construction Line > Show Bevel Angles
Toolbar..... None
Command ..... SCCLSHOWBEVEL
Permissions ... Structure > Modeling
Procedure..... None

Shows the raw bevel angles for selected construction lines using the bevel annotation text size. The raw bevel angles are the angles of the surface.



Raw Angle Definition

The angles are displayed at each point on the line. Construction lines that are restored from the database have their bevel angles display off. This command is useful as a diagnostic command to determine if the construction line has bevel angles.

SC Structure > Construction Line > Hide Bevel Angles

Button .....
Ribbon ..... Structure Modeling tab > Construction Lines > Bevel Angles drop-down > Hide Bevel Angles
Menu ..... SC Structure > Construction Line > Hide Bevel Angles
Toolbar..... None
Command ..... SCCLHIDEBEVEL
Permissions ... Structure > Modeling
Procedure..... None

Hides the raw bevel angles for selected construction lines.

SC Structure > Construction Line > Clear Bevel Angles...

Button .....
Ribbon ..... Structure Modeling tab > Construction Lines > Bevel Angles drop-down > Clear Bevel Angles
Menu ..... SC Structure > Construction Line > Clear Bevel Angles
Toolbar..... None
Command ..... SCCLCLEARBEVEL
Permissions ... Structure > Modeling
Procedure..... None

Sets all the raw bevel angles for selected construction lines to zero (a square cut). A user option exists to update or break identical construction lines. Parts that use these construction lines are updated.

SC Structure > Insert Standard Part...

Button .....
Ribbon ..... Structure Modeling tab > Parts > Insert Standard Part
Menu ..... SC Structure > Insert Standard Part
Toolbar..... Structure >
Command ..... SCINSERTSTD
Permissions ... Structure > Modeling
Procedure..... Standard Parts (page 193)

Inserts a previously-defined standard part into the drawing.

SC Structure > Convert To Structure Part

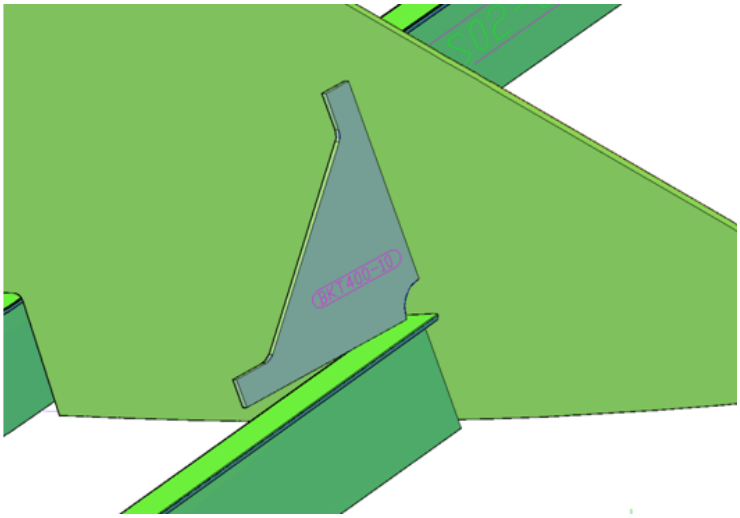
Button .....
Ribbon ..... Structure Modeling tab > Parts > Convert to Structure Part
Menu ..... SC Structure > Convert to Structure Part
Toolbar..... None
Command ..... SCCONVERTTOSTRUCTPART
Permissions ... Structure > Modeling
Procedure..... Standard Parts (page 193)

Converts a standard part to a regular part.

SC Structure > Insert Standard Part at Stiffener...

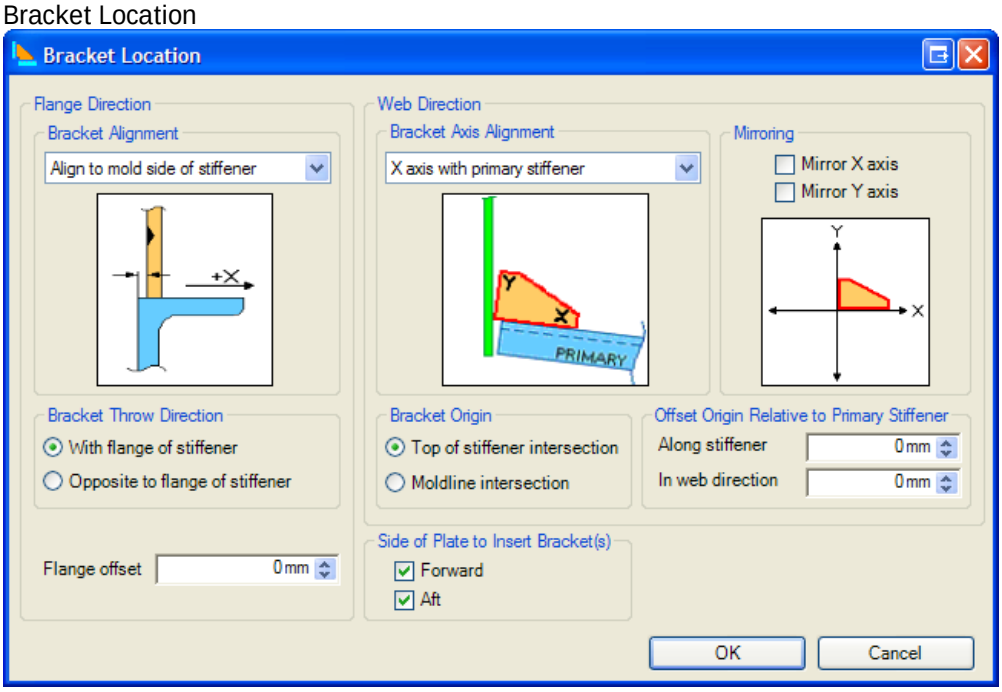
Button .....
Ribbon ..... Structure Modeling tab > Parts > Insert Standard Part drop-down > Insert Standard Part at Stiffener
Menu ..... SC Structure > Insert Standard Part at Stiffener
Toolbar..... None
Command ..... SCINSERTSTDATSTIFF
Permissions ... Structure > Modeling
Procedure..... Standard Brackets Attached to Stiffeners (page 193)

Inserts a previously-defined standard part attached to a selected stiffener. Standard bracket insertion can be tedious when there are many longitudinal stiffeners that intersect a frame and the web of the stiffeners is at different angles. This command accelerates that process by calculating the position and orientation of the standard part based on selecting a stiffener and plate or stiffener. This command will not create logical relationships between parts and only standard parts can be used.



Example case of inserted bracket

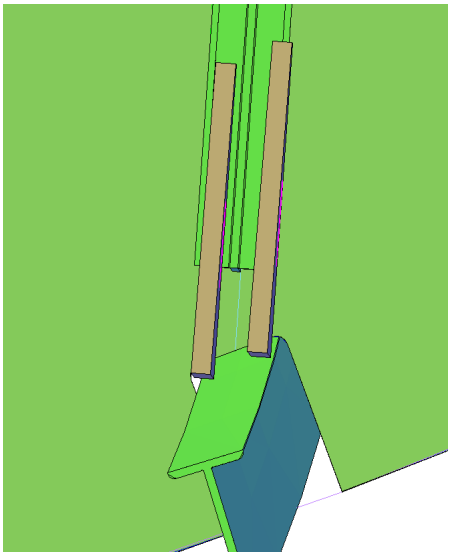
Bracket Location



Flange Direction

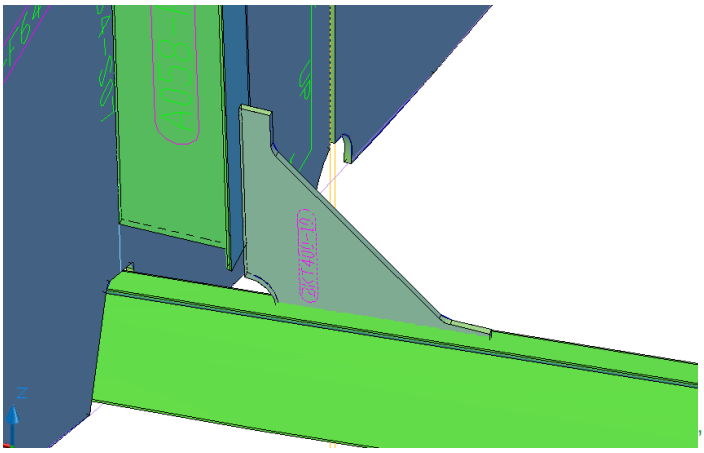
Settings pertaining to the location of the origin of the standard part only in the direction of the flange of the primary stiffener. This command will permit you to insert different instances of the same standard part and end up with two physically different parts. Or instance an asymmetrical part with a flange can have inserted instances where the flange would be up in one case and down in another.

Shifting in the flange direction is corrected so that if the bracket is not perpendicular to the face it is meant to touch (plate, top of stiffener), then the bracket is moved so that the thickness side point at the origin of the bracket isn't inside the plate or stiffener. See the following example where two brackets are shown and the one on the left is shifted down to touch the top of the stringer and the one on the right is shifted up.



Intersection with plane of stiffener flange

This does not prevent the case of an interference.



Case where bracket is interfering with the stiffener because the angle between stiffeners is acute.

Bracket Alignment

Align to mold side of stiffener – The mold plane of the standard part is aligned with the mold side of the web of the primary stiffener

Align to centers – The center of the standard part in the thickness direction is aligned with the center of the web of the primary stiffener

Align to thickness side of stiffener - The mold plane of the standard part is aligned with the mold side of the web of the primary stiffener

Bracket Throw Direction

The throw direction of the standard part.

With flange of stiffener – the throw of the standard part follows the throw of the primary stiffener

Opposite to the flange of the stiffener – the throw of the standard part follows the throw of the primary stiffener

Changing Bracket Throw Direction and Bracket Alignment changes the flange view image.

Flange offset – the distance to shift the standard part relative to the Bracket Alignment.

Web Direction

Settings pertaining to the location of the origin restricted to the plane of the web or the eventual plane of the standard part.

Bracket Axis Alignment

Align the bracket parallel to the primary stiffener or to the secondary part. X Axis refers to the X Axis in the world coordinate system when editing the standard.

Bracket Origin

Top of stiffener intersection – the origin will be located using the following rules:

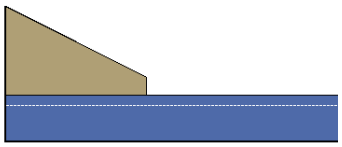
Second part selected	Origin
None	Top of the web at the end nearest to the pick point
Stiffener	The intersection of the top of the web of the primary stiffener with the plane of the flange of the secondary stiffener
Plate	The intersection of the top of the web of the primary stiffener with the plane plate

Moldline intersection – the origin will be located using the following rules:

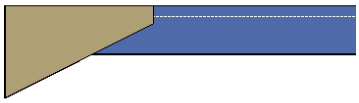
Second part selected	Origin
None	End of the moldline of the stiffener at the end nearest to the pick point
Stiffener	The intersection of the moldline of the primary stiffener with the plane of the flange of the secondary stiffener
Plate	The intersection of the moldline of the primary stiffener with the plane plate

Mirroring

Permits the mirroring of the standard part about its local axis's (the ones you see when editing the standard) for more flexible options. The standard convention is to model the bracket in the upper right quadrant in plan view.



Bracket inserted at the top of the stiffener without mirroring



Bracket inserted at the top of the stiffener with Mirror X axis checked

Offset Origin Relative to Primary Stiffener

Along stiffener– shifts the bracket in the direction of the axis of the primary stiffener. A positive value will shift it away from the second part or toward the middle of the primary stiffener if no second part selected.

In Web Direction – shifts the bracket in the direction of the web of the primary stiffener. This value would lap the primary stiffener when a negative value is entered.

Side of Plate to Insert Bracket(s)

The Plate Sides checkboxes indicate what sides of the plate to insert the bracket(s). If both checkboxes are checked then two standard parts are inserted. By default the checkboxes are checked with the sides of the plate the stiffener is on. This option will be disabled when no plate is selected.

SC Structure > Transfer Objects To Group

Button .....
Ribbon ..... Structure Modeling tab > Utilities > Transfer Objects to Group
Menu ..... SC Structure > Transfer Objects to Group
Toolbar..... None
Command ..... SCTRANSFER
Permissions ... Structure > Modeling
Procedure..... Transfer Objects to another Planar Group (page 204)

Transfers objects from the current planar group model drawings to another planar group drawing.

SC Structure > Move Planar Group

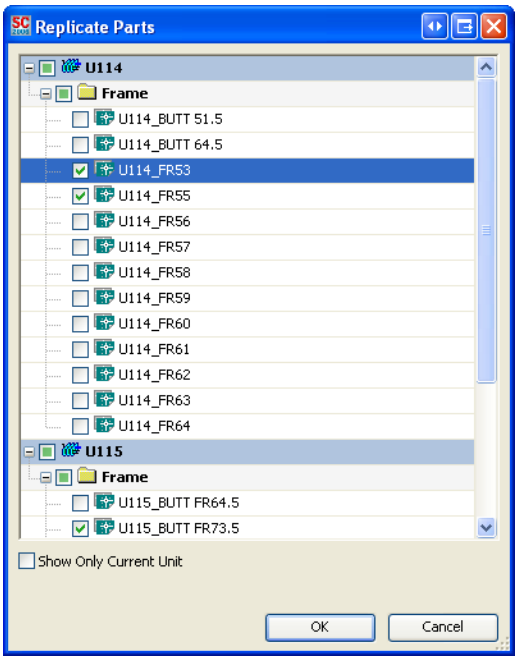
Button .....
Ribbon ..... Structure Modeling tab > Utilities > Move Planar Group
Menu ..... SC Structure > Move Planar Group
Toolbar..... Structure >
Command ..... SCMOVEPLANARGROUP
Permissions ... Structure > Modeling
Procedure..... PlanarGroupPlane Construction Line (page 153)

Moves the planar group drawing by a distance. This will move all objects in the drawing and update any related parts in other drawings.

SC Structure > Replicate Objects to Other Group

Button .....
Ribbon ..... Structure Modeling tab > Utilities > Replicate Objects to Other Groups
Menu ..... SC Structure > Replicate Objects to Other Groups
Toolbar..... None
Command ..... SCREPLICATE
Permissions ... Structure > Modeling
Procedure..... Replicate Objects to Other Planar Groups (page 203)

Replicates objects from the current planar group drawing to other parallel planar groups.



This window displays all eligible groups. Select the groups and ShipConstructor will replicate the objects. If the replicated parts are similar to the original parts and within the same unit, an identical relationship is created between the parts.

SC Structure > Check Group DWG

Button .....
Ribbon ..... Structure Modeling tab > Utilities > Check Group DWG
Menu ..... SC Structure > Check Group DWG
Toolbar..... None
Command ..... SCCHECKPGMD
Permissions ... Structure > Modeling

Checks, repairs, and logs any errors in the current planar group model drawing. An option to repair errors or log errors is presented before the check is run. If Repair Errors is not selected, any modifications listed below will not be performed. The planar group model drawing check performs the following operations:

- Check to see that all the planar group model drawings in the database exist in the appropriate directory in your project folder. If a planar group model drawing exists in the database but not in the project directory, ShipConstructor will create an empty drawing.
- Checks to see all the planar group model drawings in your project folder exist in the database. If a drawing is found to be in the project folder but not in the database, ShipConstructor will log it.
- Checks to see that all ShipConstructor layers are created and up-to-date with the database. If a layer does not exist or values are not the same as set in the database, ShipConstructor will create the layer and synchronize the layer with the database.
- Checks that all planar group planes in the database have a valid UCS associated with them. If there is no UCS that is associated with a planar group plane, ShipConstructor will create a UCS that is associated with the planar group plane. If an associated UCS does not have the same values as the planar group plane then the UCS is set to have the same value as the planar group plane stored in the database.
- Checks all structure parts:
  - Updates all structure parts from the values stored in the database (Stock, trims, piecemark, and so on).
  - Updates calculated values to the database (Weight, volume, and so on).
  - Checks and fixes structure parts integrity.

- Checks that all markings lie within the boundaries of the part that they are associated with. If any part of a mark symbol or text lies outside the part boundary, it will be noted in the generated log file.

SC Structure > Show Unused Objects

Button .....
Ribbon ..... Structure Modeling tab > Utilities > Show Unused Objects
Menu ..... SC Structure > Show Unused Objects
Toolbar..... None
Command ..... SCSHOWUNUSED
Permissions ... Structure > Modeling

Displays a list of objects in the current drawing that are not used by any parts. This tool is handy for determining extra unused lines.

SC Utilities Menu

SC Utilities > 3D Viewpoint

Button .....
Ribbon ..... Structure Modeling tab > Navigation panel > 3D Viewpoint
Menu ..... SC Utilities > 3D Viewpoint
Toolbar..... Utilities >
Command ..... SCVPOINT
Permissions ... None
Procedures..... Select a UCS Viewpoint (page 139)

Lets you select a 3D or 2D view from several pre-defined, standard views.

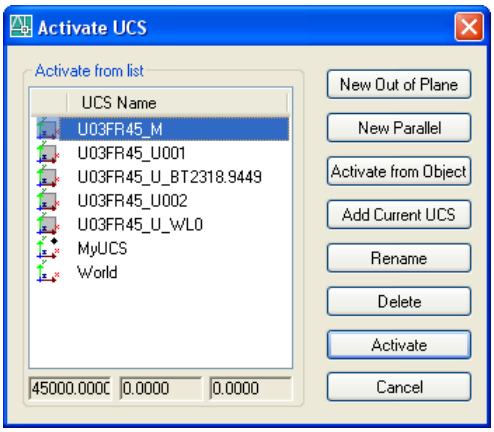
SC Utilities > Activate UCS

Button .....
Ribbon ..... Structure Modeling tab > Navigation panel > Activate UCS
Menu ..... SC Utilities > Activate UCS
Toolbar..... Utilities >
Command ..... SCUCSLIST
Permissions ... None
Procedure..... Activate a UCS (page 138)
..... Create a UCS Out of Plane with the Current UCS (page 136)
..... Create a UCS Parallel to the Current UCS (page 137)

When this command is used in the structure module, it enables the use of PlanarGroup Planes, activates a UCS (or PlanarGroup Plane), or creates a new UCS (or PlanarGroup Plane). Every PlanarGroup Plane has a UCS. However, you can make new UCSs that will not create PlanarGroup Planes.

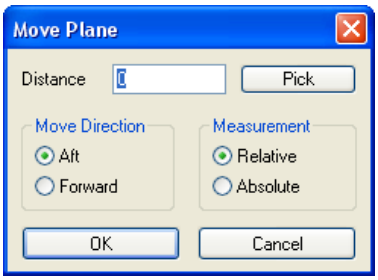
- UCS has an associated PlanarGroup Plane

- UCS only



New Out of Plane – Creates a new UCS out of plane with the current UCS. You define the new plane by picking two or three points.

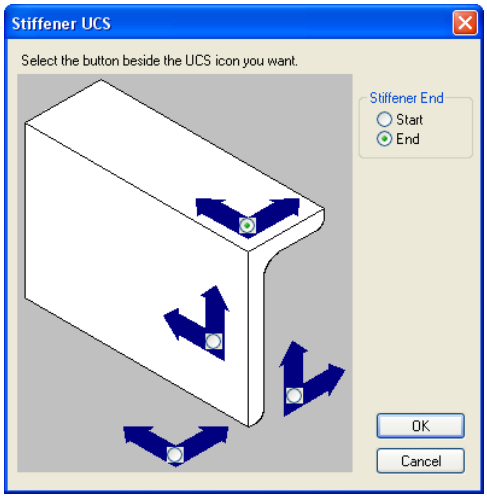
New Parallel – Creates a new UCS parallel with the current UCS.



Activate from Object – Sets a named UCS to be the current UCS or creates a UCS from a selected object. For most objects there is only one possible UCS.

Selecting a plate part brings up the following selection window. If the throw of the plate is centered, then there are three options.

Selecting a stiffener part will bring up the following selection window.



Add Currrent UCS – Adds the currently selected UCS to the database.

Rename – Allows you to rename the selected UCS in the list. For Planar Group Planes, you cannot rename the prefix.

Delete – Deletes the selected UCS in the list.

Activate – Activates the selected UCS in the list.

SC Utilities > UCS > Flip UCS X

Button .....
Ribbon ..... Structure Modeling tab > Modeling/Production Utilities panel > Flip UCS X > Flip UCS X
Menu ..... SC Utilities > UCS > Flip UCS X
Toolbar..... Assembly >
Command ..... SCFLIPUCS
Permissions ... None
Procedure..... None

Changes the current UCS so that the X Axis is pointing towards the –X Axis direction.

SC Utilities > UCS > Flip UCS Y

Button .....
Ribbon ..... Structure Modeling tab > Modeling/Production Utilities panel > Flip UCS X > Flip UCS Y
Menu ..... SC Utilities > UCS > Flip UCS Y
Toolbar..... Assembly >
Command ..... SCFLIPUCSY
Permissions ... None
Procedure..... None

Changes the current UCS so that the Y Axis is pointing towards the –Y Axis direction.

SC Utilities > UCS > Swap UCS XY Axis

Button .....
Ribbon ..... Structure Modeling tab > Modeling/Production Utilities panel > Flip UCS X > Swap UCS XY Axis
Menu ..... SC Utilities > UCS > Swap UCS XY Axis
Toolbar..... Assembly >
Command ..... SCSWAPUCSXY
Permissions ... None
Procedure..... None

Changes the current UCS so that the X Axis is pointing towards the Y Axis direction and the Y Axis is pointing towards the X Axis direction.

SC Utilities > Hide Objects

Button .....
Ribbon ..... Structure Modeling tab > Navigation panel > Hide Objects
Menu ..... SC Utilities > Hide Objects
Toolbar..... Utilities >
Command ..... SCHIDE
Permissions ... None
Procedure..... Hide or Show Objects (page 141)

Hides the objects selected. Hidden objects are remembered in the order you hid them while you are in the current drawing.

SC Utilities > Unhide Objects

Button .....
Ribbon ..... Structure Modeling tab > Navigation panel > Unhide Objects > Unhide Objects
Menu ..... SC Utilities > Unhide Objects
Toolbar..... Utilities >
Command ..... SCUNHIDE
Permissions ... None
Procedure..... Hide or Show Objects (page 141)

Shows hidden objects (SC Utilities > Hide Objects (page 454)) in the reverse order that you hid them.

SC Utilities > Unhide All Objects

Button .....
Ribbon ..... Structure Modeling tab > Navigation panel > Unhide Objects > Unhide All Objects
Menu ..... SC Utilities > Unhide All Objects
Toolbar..... None
Command ..... SCUNHIDEALL
Permissions ... None
Procedure..... Hide or Show Objects (page 141)

Shows all hidden objects (SC Utilities > Hide Objects (page 454)).

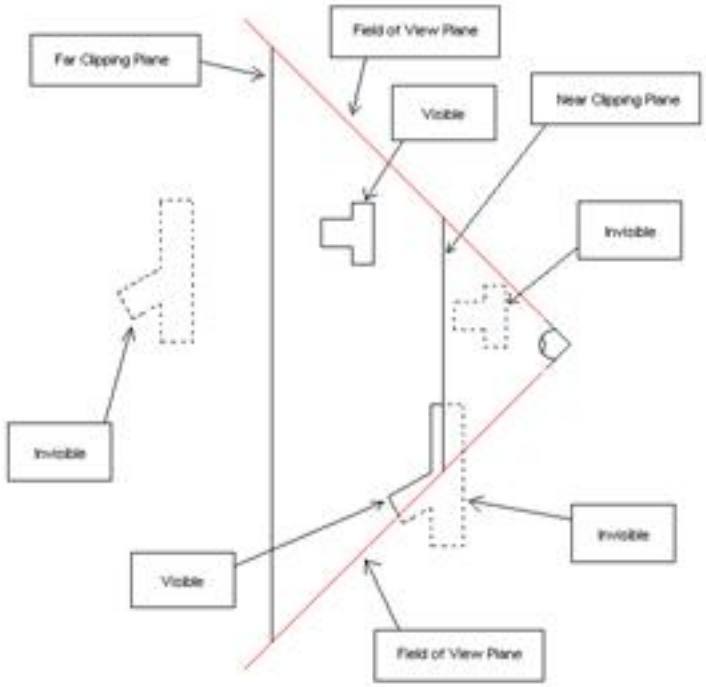
Note: When determining if an object is not visible in the drawing, run this command and turn on and thaw all layers.

SC Utilities > Clip Current View

Button .....
Ribbon ..... Structure Production tab > Modeling/Production Utilities panel > Clip Current View
Menu ..... SC Utilities > Clip Current View
Toolbar..... None
Command ..... SCCLIPVIEW
Permissions ... None

Clip Current View is a utility function that provides a convenient method for setting up AutoCAD clipping planes inside a viewport.

Note: Before running the command, you need to switch to the viewport in which the clipping plane is to be applied.



1. The command line prompts you to:

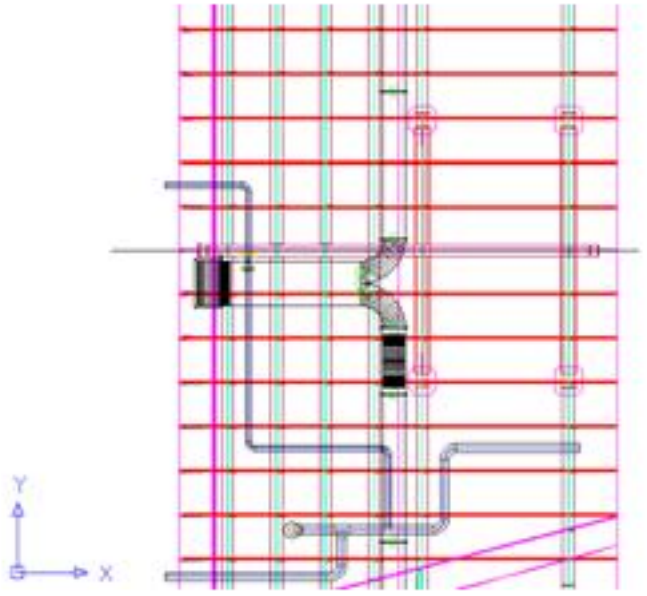
```
Enter Point on Plane 1:
```

2. Choose a point on one of the clipping planes to be used (whether it is the front or back clipping plane will be determined by ShipConstructor).
3. The command line prompts you to enter a second point:

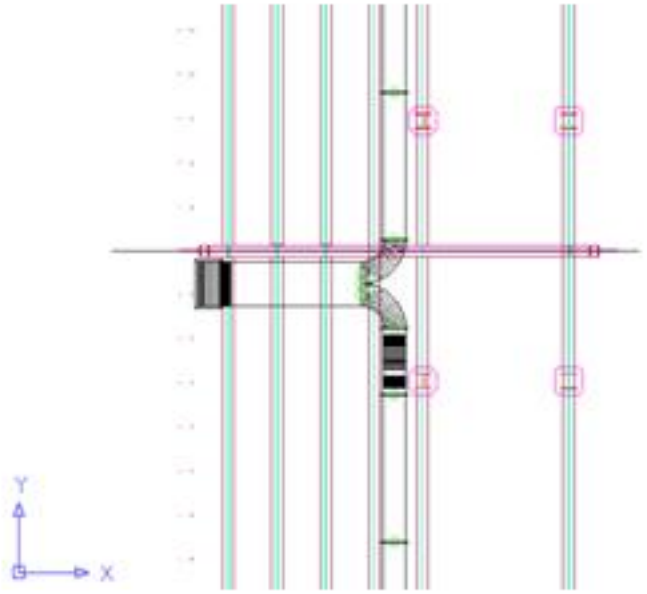
```
Enter Point on plane 2:
```

4. Choose a point on the second clipping plane (whether it is the front or back clipping plane will be determined by ShipConstructor).

Note: If the clipping planes are not to your liking you can remove them with the command SC Utilities > Remove Clip (see below).



Before clipping.



After clipping.

SC Utilities > Remove Clip

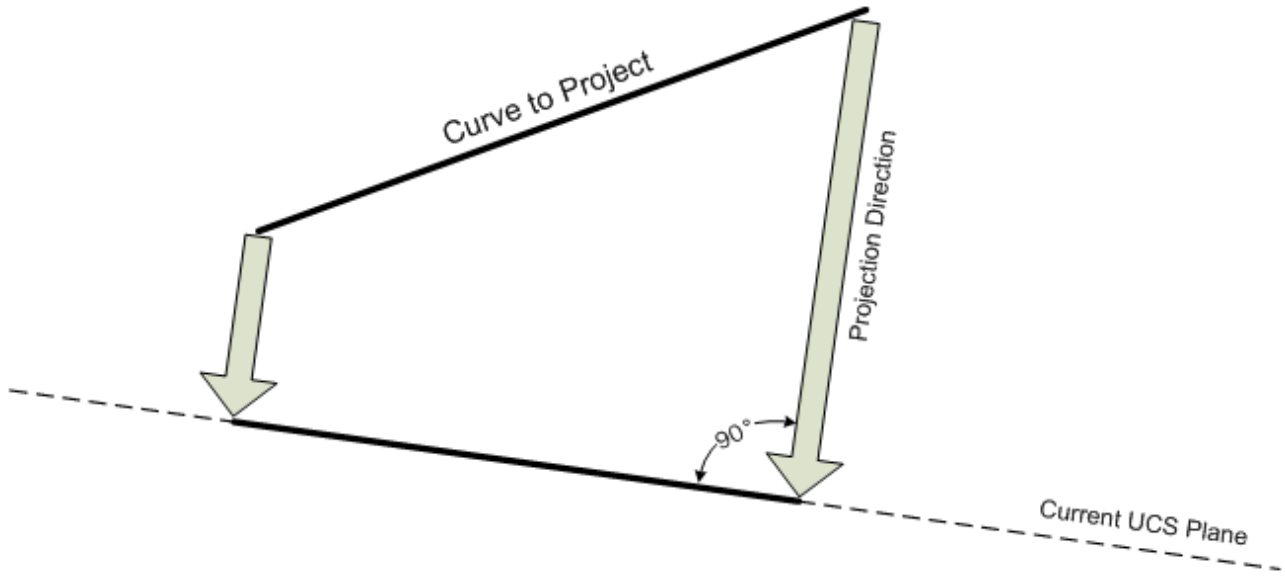
Button .....
Ribbon ..... Structure Production tab > Modeling/Production Utilities panel > Remove Clip
Menu ..... SC Utilities > Remove Clip
Toolbar..... None
Command ..... SCCLEARCLIP
Permissions ... None

This command removes the AutoCAD clipping planes (SC Utilities > Clip Current View (page 455)) from the currently selected viewport.

SC Utilities > 3D to 2D

Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities panel > 3D to 2D
Menu ..... SC Utilities > 3D to 2D
Toolbar..... Structural Tools >
Command ..... SCCONV
Permissions ... None
Procedure..... Convert a 3D Object to a 2D Object (page 207)

Converts a 3D object to a 2D object on the XY plane of the current UCS.



The direction of projection for the 3D to 2D command

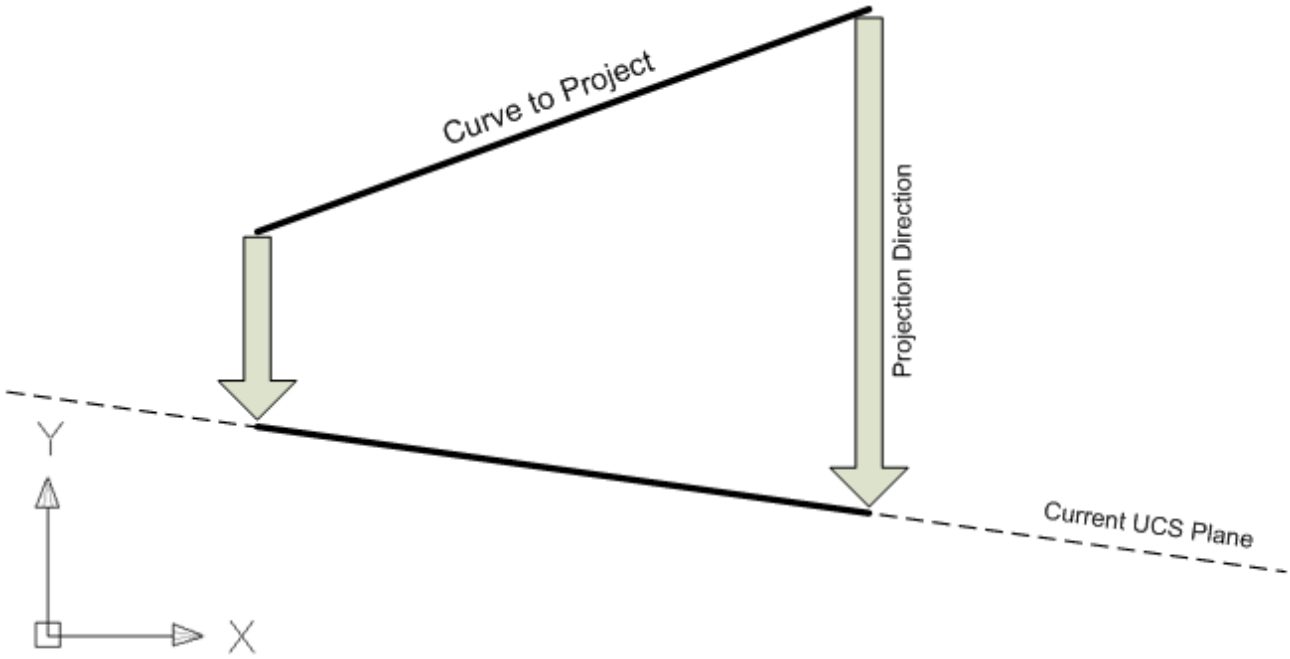
This command works on the following object types:

- Lines
- Arcs
- Circles
- Polylines
- 3D Polylines
- Text, MText
- Points
- Solids (not 3D solids)
- User Construction Lines

SC Utilities > Orthographic Projection

Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities panel > Orthographic Projection
Menu ..... SC Utilities > Orthographic Projection
Toolbar..... None
Command ..... SCPROJECTONTOUCS
Permissions ... None
Procedure..... Project Shapes (page 207)

Projects a shape onto the current UCS. You choose the direction to project from one of the three orthogonal world directions (X, Y, or Z).



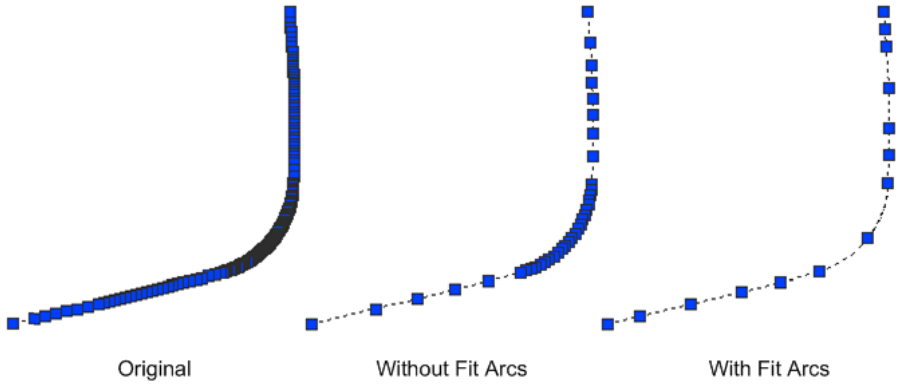
The direction of projection for the orthographic projection command

This command only works on lines.

SC Utilities > Remove Vertices Below Tolerance

Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities panel > Remove Vertices Below Tolerance
Menu ..... SC Utilities > Remove Vertices Below Tolerance
Toolbar..... None
Command ..... SCBELOWTOL
Permissions ... None
Procedure..... Reduce the Number of Vertices on Polylines (page 207)

Reduces the number of vertices on a polyline by removing vertices within a given tolerance and converting the segment to a straight line. If line segments form an arc and are within the tolerance, an arc is inserted into the polyline.



This command is helpful in reducing the complexity of some curves or sections that will be used in the model. Since the NC machine cannot usually cut any more accurate than 0.01, there is no need to have a model more accurate.

SC Utilities > Convert Ellipse/Spline to Polyline

Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities panel > Convert Ellipse/Spline to Polyline
Menu ..... SC Utilities > Convert Ellipse/Spline to Polyline
Toolbar..... None
Command ..... SCCONVELLIPSE
Permissions ... None
Procedure..... Convert an Ellipse or Spline to a Polyline (page 208)

Converts an ellipse, arc, circle, or spline to a polyline with only straight line segments. The maximum deviation from the original curve is 0.01 AutoCAD units. To use a lower resolution run SC Utilities > Remove Vertices Below Tolerance.

SC Utilities > Layer > Activate

Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities panel > Activate > Activate
Menu ..... SC Utilities > Layer > Activate
Toolbar..... Utilities > Layers Flyout >
Command ..... SCACTLAYER
Permissions ... None
Procedure..... Activate a Layer (page 140)

Turns the selected layer on and thaws it if it is frozen.

SC Utilities > Layer > Deactivate

Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities panel > Activate > Deactivate
Menu ..... SC Utilities > Layer > Deactivate
Toolbar..... Utilities > Layers Flyout >
Command ..... SCDELAYER
Permissions ... None
Procedure..... Deactivate a Layer (page 141)

Turns the selected layer off and freezes it.

SC Utilities > Layer > Copy Geometry to

Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities panel > Activate > Copy Geometry to
Menu ..... SC Utilities > Layer > Copy Geometry to
Toolbar..... Utilities > Layers Flyout >
Command ..... SCCOPYGEO
Permissions ... None
Procedure..... Copy Objects from One Layer to Another (page 141)

Copies objects to the selected layer. Objects inside blocks are also moved to the destination layer.

SC Utilities > Layer > Move Geometry to

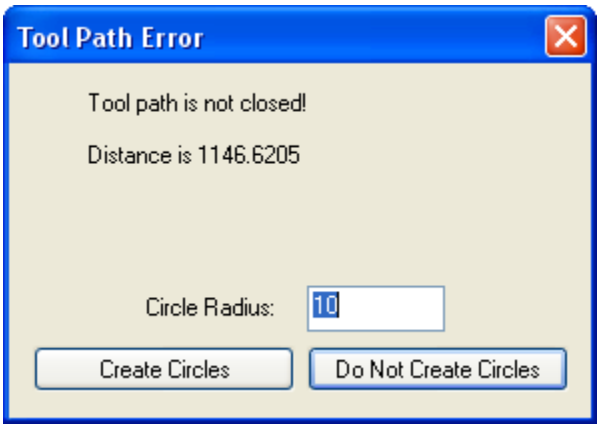
Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities panel > Activate > Move Geometry to
Menu ..... SC Utilities > Layer > Move Geometry to
Toolbar..... Utilities > Layers Flyout >
Command ..... SCMOVEGEO
Permissions ... None
Procedure..... Move Objects from One Layer to Another (page 141)

Moves objects from one layer to another. If you move a block, the objects inside the block are also moved to the destination layer.

SC Utilities > Toolpath

Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities panel > Toolpath
Menu ..... SC Utilities > Toolpath
Toolbar..... Utilities >
Command ..... SCTOOLPATH
Permissions ... None
Procedure..... Create a Closed Toolpath (page 206)

Creates a closed polyline (when possible) from lines that have no gaps. If the connected path is not closed, you will see the following warning. Duplicate lines will generate warnings that allow you to remove them.

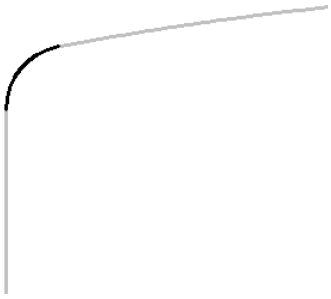


This command uses the Project Settings > Structure > Toolpath Gap Tolerance to close small gaps. The gap is closed by

SC Utilities > Fillet

Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities panel > Fillet
Menu ..... SC Utilities > Fillet
Toolbar..... Utilities >
Command ..... SCTOOLPATH
Permissions ... None
Procedure..... None

Creates a fillet between two curves. AutoCAD's fillet command can fail to generate a result when two complex polylines are filleted.



SC Utilities > Mirror about Centerline

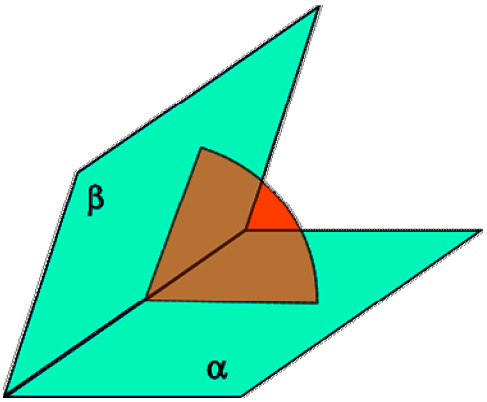
Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities panel > Mirror about Centerline
Menu ..... SC Utilities > Mirror about Centerline
Toolbar.....
Command ..... SCMIRRORCL
Permissions ... None
Procedure..... None

Makes a copy of the selected objects and mirrors them about the vessel's centerline.

SC Utilities > Dihedral Angle

Button .....
Ribbon ..... Structure Modeling tab > Utilities panel > Dihedral Angle
Menu ..... SC Utilities > Dihedral Angle
Toolbar.....
Command ..... SCDHANGLE
Permissions ... None
Procedure..... Find the Dihedral Angle between Two Planar Objects (page 206)

Determines the dihedral angle between two planar objects



This command works on the following object types:

- Plates
- Stiffeners with a straight construction line
- Faceplates with a straight construction line
- Any object that is planar

SC Utilities > Reload Drawing

Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities panel > Reload Drawing
Menu ..... SC Utilities > Reload Drawing

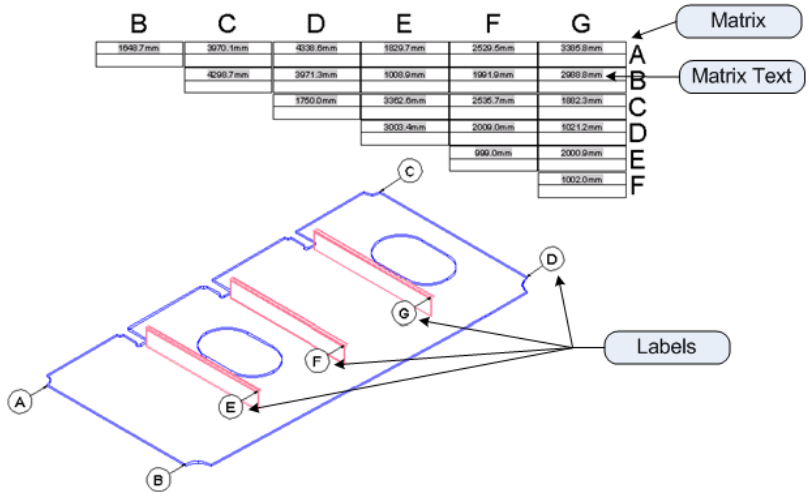
Toolbar..... Utilities >
Command ..... SCRELOAD
Permissions ... None
Procedure..... None

Updates all parts and construction lines in the drawing to reflect what is stored in the database. This command simulates the closing and opening of the current drawing. You cannot undo this command. Any objects that were erased will be removed from the database.

SC Utilities > Create Quality Matrix

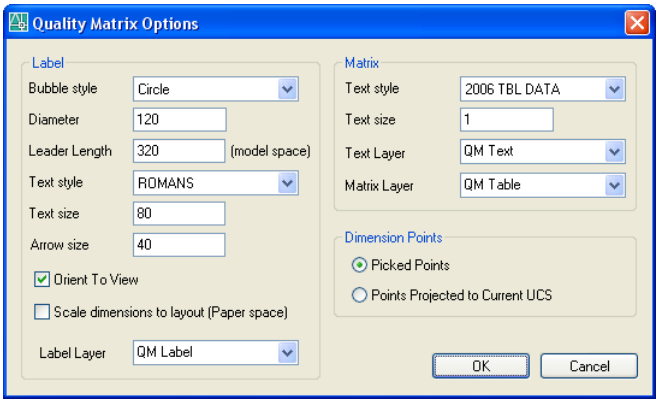
Button .....
Ribbon ..... Structure Production > Modeling/Production Utilities tab > Create Quality Matrix
Menu ..... SC Utilities > Create Quality Matrix
Toolbar..... None
Command ..... SCQUALMTX
Permissions ... None
Procedure..... Insert a Quality Control Matrix into an Assembly Drawing (page 237)

Inserts a quality control matrix into a drawing. A quality control matrix is used for dimensional checking. It shows the straight-line distances between selected points.



After inserting the matrix, any of the dimension points or labels can be moved. The length values will be updated when a View > Regen is run. The format of the lengths in the matrix is controlled by the Format > Units settings.

Quality Matrix Options



Label

Bubble style – The style of the bubble surrounding the letter label.

Bubble Size/Bubble Margin/Diameter – The size of the bubble. For the Bubble and Rectangle styles this is a margin between the text and the bubble. If Scale dimensions to layout is checked then this value is in paperspace units; otherwise it is in model space units.

Leader Length – The length of the leader line between the picked dimension point and the label.

Text style – The style of the text within the label.

Text size – The size of the text within the label. If Scale dimensions to layout is checked then this value is in paperspace units; otherwise it is in model space units.

Arrow size – The size of the arrow. Setting this value to zero will suppress the displaying of an arrow. If Scale dimensions to layout is checked then this value is in paperspace units; otherwise it is in model space units.

Orient To View – Orients the label to be facing you when you are looking from any viewpoint.

Scale dimensions to layout (Paper space) – If enabled, the labels will be scaled to paperspace; otherwise model space units are used.

Label Layer – Layer to place all the labels on.

Matrix

Text style – The text style of the text in the matrix of dimensions. This lists all the text styles in the current drawing.

Text size – The size of the text for the matrix of dimensions. This also determines the size of the matrix. If you started the command while in a layout then this value will be in paperspace units.

Text Layer – Layer to place the dimensions on.

Matrix Layer – Layer to place the matrix and column headers on

Dimension Points

Picked Points – When picking points, ShipConstructor uses that point as the dimension point.

Points Projected to Current UCS – When picking points, ShipConstructor creates a dashed line from that point to a point on the current UCS and dimensions from that point. This makes it easier to pick points, since you do not have to zoom in close and make sure you are picking points on the front of the plate.

SC Utilities > Property Labels > Property Label

Button .....
Ribbon ..... Structure Production tab > Property Label panel > Property Label
Menu ..... SC Utilities > Property Labels > Property Label
Toolbar..... None
Command ..... SCOBJFIELDLABEL
Permissions ... Edit production drawing if in a production drawing or edit model drawing if in model drawing
Procedure..... Inserts a ShipConstructor field label into a drawing

SC Utilities > Property Labels > Copy Field Label

Button .....
Ribbon ..... Structure Production tab > Property Label panel > Copy Field Label
Menu ..... SC Utilities > Property Labels > Copy Field Label
Toolbar..... None
Command ..... SCFIELDLABELCOPY
Permissions ... Edit production drawing if in a production drawing or edit model drawing if in model drawing
Procedure..... Copies mtext block and creates new leader. Also allows user to select new item to change the object references within the fields.

SC Utilities > Property Labels > Copy Field Label Quick

Button .....
Ribbon ..... Structure Production tab > Property Label panel > Copy Field Label Quick
Menu ..... SC Utilities > Property Labels > Copy Field Label Quick
Toolbar..... None
Command ..... SCFIELDLABELCOPYQUICK
Permissions ... Edit production drawing if in a production drawing or edit model drawing if in model drawing
Procedure..... Copies mtext block and creates new leader using original leader geometry. Also allows user to select new item to change the object references within the fields.

SC Utilities > Property Labels > Edit Field Label

Button .....
Ribbon ..... Structure Production tab > Property Label panel > Edit Field Label
Menu ..... SC Utilities > Property Labels > Edit Field Label
Toolbar..... None
Command ..... SCEDITFIELD
Permissions ... Edit production drawing if in a production drawing or edit model drawing if in model drawing
Procedure..... Select mtext and edit the text and fields.

SC Utilities > Property Labels > Replace Object References

Button .....
Ribbon ..... Structure Production tab > Property Label panel > Replace Object References
Menu ..... SC Utilities > Property Labels > Replace Object References
Toolbar..... None

Command ..... SCREPLACEOBJREFINFIELD
Permissions ... Edit production drawing if in a production drawing or edit model drawing if in model drawing
Procedure..... Select mtext and then select new object to point the fields to.

SC Utilities > Property Labels > Set Single Click Field Label

Button .....
Ribbon ..... Structure Production tab > Property Label panel > Set Single Click Field Label
Menu ..... SC Utilities > Property Labels > Set Single Click Field Label
Toolbar..... None
Command ..... SCSETSINGLECLICKFIELDLABEL
Permissions ... Edit production drawing if in a production drawing or edit model drawing if in model drawing
Procedure..... Toggles the ShipConstructor variable that determines whether first leader pick point is acquired by when the object is selected, or whether it is determined by separate point selection.

SC Utilities > List Item within Block/Xref

Button .....
Ribbon ..... Structure Production tab > Modeling/Production Utilities panel > List item within Block/Xref
Menu ..... SC Utilities > List item within Block/Xref
Toolbar..... None
Command ..... SCLISTSUBENT
Permissions ... None
Procedure..... Display Information for an Object within an MLinked Drawing (page 142)

Displays LIST information for an object within an mlinked or other XREFed drawing.

SC Utilities > Check Local Interferences

Button .....
Ribbon ..... Structure Production tab > Modeling/Production Utilities panel > Check Local Interferences
Menu ..... SC Utilities > Check Local Interferences
Toolbar..... None
Command ..... SCINTERCHECK
Permissions ... None
Procedure..... Check Local Interferences (page 203)

Checks for collisions between objects in a single drawing without having to create an Interference drawing.

SC Utilities > Export > Export to DWG...

Button .....
Ribbon ..... Structure Production tab > Modeling/Production Utilities panel > Export to DWG...
Menu ..... SC Utilities > Export to DWG...
Toolbar..... None
Command ..... SCEXPORTDWG
Permissions ... None
Procedure..... Export a ShipConstructor Drawing to an AutoCAD only Drawing (page 143)

Exports the current drawing as an AutoCAD only drawing. The entire drawing will be duplicated with only native AutoCAD entities, including any hidden objects or layers.

Note: The export does not take the Structural Display Options into account; the solids, piecemarks and production info of the parts will all be visible in the exported drawing regardless of the Structural Display Options, (similar to how the AutoCAD **EXPLODE** command works for parts). See SC Structure > Show / Hide Options (page 394) for more information on the Structural Display Options.

SC Utilities > Export > Export to NWD

Button .....
Ribbon ..... Structure Production tab > Modeling/Production Utilities panel > Export to NWD
Menu ..... SC Utilities > Export > Export to DWG...
Toolbar..... None
Command ..... SCNWDOUT
Permissions ... None
Procedure..... Export ShipConstructor Drawings to NavisWorks Drawings (page 144)

This command has been deprecated and is no longer supported.

SC Utilities > Export > Export to NWC

Button .....
Ribbon ..... Structure Production tab > Modeling/Production Utilities panel > Export to NWC
Menu ..... SC Utilities > Export > Export to DWG...
Toolbar..... None
Command ..... SCNWCOUT
Permissions ... None
Procedure..... Export ShipConstructor Drawings to NavisWorks Drawings (page 144)

Exports selected drawings as separate Navisworks NWC files. The resulting files are placed beside the original drawing with the same filename with extension NWC. The export settings are used from the NWOPT settings.

SC Utilities > Snap

Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities panel > Snap
Menu ..... SC Utilities > Snap
Toolbar..... Utilities >
Command ..... None
Permissions ... None
Procedure..... None

Opens the AutoCAD Drafting Settings window with the ShipCon Snap tab selected, letting you set the ShipConstructor snap options.

SC Utilities > Random Color

Button .....
Ribbon ..... Structure Modeling tab > Modeling Utilities panel > Random Color
Menu ..... SC Utilities > Random Color
Toolbar..... None
Command ..... SCRANDCOLOR
Permissions ... None
Procedure..... None

Assigns a random color to each of the selected entities in the current drawing.

SC Utilities > ShipCAM > Import Files

Button ..... None
Ribbon ..... None
Menu ..... SC Utilities > ShipCAM > Import Files
Toolbar..... None
Command ..... SCIN
Permissions ... None
Procedure..... Import a Hull Model from ShipCAM (page 126)

Lets you import a 3D wireframe model from ShipCAM into the current drawing. This utility function can be run in plain AutoCAD.

SC Utilities > ShipCAM > Convert IGES to Mesh

Button ..... None
Ribbon ..... None
Menu ..... SC Utilities > ShipCAM > Convert IGES to Mesh
Toolbar..... None
Command IGES2MSH
Permissions ... None
Procedure..... None

Converts an IGES file to a ShipCAM MSH file. This function is made available because of issues with ShipCAM import of IGES files. This utility function can be run in plain AutoCAD.

Command Line Commands

Plate Part Extract Polyline

Menu ..... None
Toolbar..... None
Command ..... SCEXTRACTOUTSIDE
Permissions ... None
Procedure..... None

Extracts a copy of the outside toolpath of a plate part and creates a polyline on the current layer.

Create Boundary Polyline

Menu ..... None
Toolbar..... None
Command ..... SCBO
Permissions ... None
Procedure..... None

Creates a closed polyline using the same algorithm as creating a plate part. This command has a similar effect as AutoCAD's **BOUNDARY** command.

Create Boundary Polyline Debug

Menu ..... None
Toolbar..... None
Command ..... SCBO
Permissions ... None
Procedure..... None

A diagnostic version of the boundary command that places text objects at the intersections it visits. The text object is labeled by increasing numbers (1, 2, 3). ShipConstructor goes vertically up until it intersects a curve and then tries to follow the leftmost path until it comes back to the start.

Create Boundary Polyline from Outside Pickpoint

Menu ..... None
Toolbar..... None
Command ..... SCBP
Permissions ... None
Procedure..... None

Creates a polyline from a point picked below the eventual closed polyline. ShipConstructor goes vertically up until it intersects a curve and then tries to follow the rightmost path until it comes back to the start. This command can sometimes be useful to create closed polylines for openings in plate parts.

Show Plate Part Construction Lines

Menu ..... None
Toolbar..... None
Command ..... SCSHOWPPCLS
Permissions ... Structure > Modeling
Procedure..... None

Shows (unhides) the construction lines associated with the selected plate parts. If the construction lines are already visible then no difference is shown.

Hide Plate Part Construction Lines

Menu ..... None
Toolbar..... None
Command ..... SCHIDEPPCLS
Permissions ... Structure > Modeling
Procedure..... None

Hides the construction lines associated with the selected plate parts. If the construction lines are already invisible then no difference is shown. SC Utilities > Unhide All Objects will show these hidden lines again.

Show Plate Part Construction Lines Only

Menu ..... None
Toolbar..... None
Command ..... SCSHOWONLYPPCLS
Permissions ... Structure > Modeling
Procedure..... None

Shows (unhides) the construction lines associated with the selected plate parts and hides all other objects. SC Utilities > Unhide All Objects (page 454) will show the hidden objects again.

Show Plate Part and Its Construction Lines Only

Menu ..... None
Toolbar..... None
Command ..... SCSHOWONLYPPANDCLS
Permissions ... Structure > Modeling
Procedure..... None

Shows (unhides) the construction lines associated with the selected plate parts and the selected plate parts and hides all other objects. SC Utilities > Unhide All Objects (page 454) will show the hidden objects again.

Show Extended Data

Menu ..... SC Utilities > Show Extended Data
Toolbar..... None
Command ..... SCXDATA
Permissions ... None
Procedure..... None

Shows the hidden ShipConstructor information that is tagged to non-ShipConstructor objects. Only a few objects have extended data attached to them.

Fix Identical Part Names

Menu ..... None
Toolbar..... None
Command ..... SCFIXIDENTPARTNAMES
Permissions ... None
Procedure..... None

This will use the part name of the selected parts (generated or non-generated) as the basis for the part names of all parts with identical relationships to the selected parts.

Menu ..... None
Toolbar..... None
Command ..... SCGENERATEIDENTPARTNAMES
Permissions ... None
Procedure..... None

This will find the generated part name with the lowest auto-number for each selected part and their identicals and use this auto-number to regenerate the part names for all parts identical to the parts selected. Parts that have non-generated names will be changed to have generated names.

Clean Geometry of all Twisted Stiffeners

Menu ..... None
Toolbar..... None
Command ..... SCCLEANAIITWISTEDSTIFFENERS
Permissions ... None
Procedure..... None

This command will run a cleaning procedure on all the twisted stiffener parts in the current project. The purpose of this operation is to allow users to correct issues that can arise from the complexity of the twisted stiffener geometry. This operation may remove points from the parts underlying geometry so users should only use it if they need to. Under normal circumstances there should be no need to use this operation. Possible uses for this operation are:

- To reduce the density of points in the twisted stiffener model to improve performance and reduce complexity.
- To eliminate abnormalities in the geometry which are causing the part to model or plate nest incorrectly.

Set Dynamic Marking Block Text Static

Menu ..... None
Toolbar..... None
Command ..... SCSETDMBTEXTSTATIC
Permissions ... None
Procedure..... None

This command is only available in a dynamic marking block standard drawing, accessible through the dynamic marking block catalog (see Dynamic Marking Block Standards (page 101)). This command sets a text object to static. Static text cannot be changed on individual instances of a DMB.

Set Dynamic Marking Block Text Normal

Menu ..... None
Toolbar..... None
Command ..... SCSETDMBTEXTNORMAL
Permissions ... None
Procedure..... None

This command is only available in a dynamic marking block standard drawing, accessible through the dynamic marking block catalog (see Dynamic Marking Block Standards (page 101)). Normal text (dynamic text) can be changed on individual instances of a DMB.

Copy Product Hierarchy

Menu ..... None
Toolbar..... None
Command ..... -SCProductHierarchyCopy
Permissions ... General > Product Hierarchy > Product Hierarchy – Edit
Procedure..... None

This command creates a copy of an existing product hierarchy. You may specify if you want to copy either the Assembly hierarchy or Assembly and parts hierarchy.

Export Product Hierarchy

Menu ..... None
Toolbar..... None
Command ..... -SCProductHierarchyExport
Permissions ... None
Procedure..... None

This command exports an existing product hierarchy to an xml file.

Import Product Hierarchy

Menu ..... None
Toolbar..... None
Command ..... -SCProductHierarchyImport
Permissions ... General > Product Hierarchy > Product Hierarchy – Edit
Procedure..... None

This command imports a product hierarchy from an xml file that has been generated by the Export Product Hierarchy command.

Part View Load via Extents – SCPartViewLoadByExtents

Menu ..... None
Toolbar..... None
Command ..... SCPartViewLoadByExtents
Permissions ... none
Procedure..... None

Loads all supported ShipConstructor parts which reside in or intersect the specified WCS extents. The jig for the extents selection works best if WCS is your current UCS and you are in an isometric view. You may also type the desire extents in on the command line. This command should work in all ShipConstructor drawings

Part View Load via Selected Object Extents – SCPartViewLoadBySelectedExtents

Menu ..... None
Toolbar..... None
Command ..... SCPartViewLoadBySelectedExtents
Permissions ... none
Procedure..... None

Similar to Load via Extents, this command takes a selection set as input and loads all supported ShipConstructor parts which reside in or intersect the specified WCS extents of the objects in the selection set.

Part View Load Associated Structure Parts – SCPartViewLoadAssociated

Menu ..... None
Toolbar..... None
Command ..... SCPartViewLoadAssociated
Permissions ... none
Procedure..... None

Loads all parts that are related to the selected parts (directly or indirectly) via a mirror or an identical relationship. This command only works within a structure model drawing.

Part View Delete All Part Views in Drawing – SCPartViewDeleteAll

Menu ..... None
Toolbar..... None
Command ..... SCPartViewDeleteAll
Permissions ... none
Procedure..... None

Deletes all Part Views in the current drawing.

Diagnostic Commands

Command Trace - Enable

Menu ..... None
Toolbar..... None
Command ..... SCENABLECMDT
Permissions ... None
Procedure..... None

Records all the commands that are run and saves them to a log file. This is sometimes useful in recording a long process of commands that help support staff reproduce a problem a user is encountering.

The log files are saved to the <project>\TraceLogs folder. These log files are ASCII text files that can be viewed with any text file viewer.

Command Trace - Disable

Menu ..... None
Toolbar..... None
Command ..... SCDISABLECMDT
Permissions ... None
Procedure..... None

Turns off the recording of commands. Recording is otherwise stopped when ShipConstructor is closed.

Debug Trace - Enable

Menu ..... None
Toolbar..... None
Command ..... SCENABLEDT
Permissions ... None
Procedure..... None

Turns on the showing of internal warnings or errors to the AutoCAD command line. You should not need to use this command unless instructed to do so.

Debug Trace - Log

Menu ..... None
Toolbar..... None
Command ..... SCENABLEDTLOG
Permissions ... None
Procedure..... None

Generates a log file of the internal warnings or errors. The warnings are also displayed on the AutoCAD command line. The log files are saved to the <project>\TraceLogs folder. These log files are ASCII text files that can be viewed with any text file viewer.

Debug Trace - Disable

Menu ..... None
Toolbar..... None
Command ..... SCENABLEDT
Permissions ... None
Procedure..... None

Stops the debugging mode. See Debug Trace - Enable (page 473).

Performance Monitor - Start

Menu ..... None
Toolbar..... None
Command ..... SCSPM
Permissions ... None
Procedure..... None

Starts the performance monitoring diagnostic tool. Performance monitoring is an internal command that aids in determining the bottlenecks. You should not need to use this command unless instructed to do so.

Performance Monitor - End

Menu ..... None
Toolbar..... None
Command ..... SCEPM
Permissions ... None
Procedure..... None

Stops the performance monitoring. See Performance Monitor - Start (page 474).

Performance Monitor - Display

Menu ..... None
Toolbar..... None
Command ..... SCDPM
Permissions ... None
Procedure..... None

Outputs the performance monitor data to an Excel spreadsheet and opens Excel.

Save Structure Drawing State

Menu ..... None
Toolbar..... None
Command ..... SCSAVEIMAGE
Permissions ... None
Procedure..... None

This command is helpful for our support staff to see the exact state that you are in a drawing when you encounter an issue. This command saves a drawing called DataManagerImage.dwg that you can transmit to our support staff to diagnose a problem.

Load Structure Drawing State from File

Menu ..... None
Toolbar..... None
Command ..... SCLOADIMAGE
Permissions ... None
Procedure..... None

This command is used in conjunction with Save Structure Drawing State (page 474). You should not need to use this command.

DataManager Dump

Menu ..... None
Toolbar..... None
Command ..... SCDUMPDM
Permissions ... None
Procedure..... None

This is an internal command that outputs cached (in memory) data to an XML file. You should not need to use this command.

Appendix B: Structure Production Command Reference

Toolbars

Assembly Toolbar

Product Hierarchy

See SC Product Hierarchy > Develop Product Hierarchy (page 508)

See SC Utilities > UCS > Flip UCS X (page 453)

Flip UCS Y

See SC Utilities > UCS > Flip UCS Y (page 453)

Swap UCS XY Axis

See SC Utilities > UCS > Swap UCS XY Axis (page 454)

Orientation Icon

See SC Assembly > Orientation Icon (page 490)

Manual Label from BOM

See SC Assembly > Label > Manual Label from BOM (page 496)

Copy Label from BOM

See SC Assembly > Label > Copy Label from BOM (page 496)

Label on Visible Edge On/Off

See SC Assembly > Label > Label on Visible Edge On/Off (page 497)

Curved Plates in Visible Edge Detection On/Off

See SC Assembly > Label > Curved Plates in Visible Edge Detection On/Off (page 497)

Adjacent Part Edge Tolerance

See SC Assembly > Label > Adjacent Part Edge Tolerance (page 498)

Edge Length Filter

See SC Assembly > Label > Edge Length Filter (page 498)

Edge Determination Minimum Angle

See SC Assembly > Label > Edge Determination Minimum Angle (page 498)

Label Reset Automatic Settings

See SC Assembly > Label > Label Reset Automatic Settings (page 498)

Leader Utils Toolbar (page **Error! Bookmark not defined.**)

Create Leader Distribution Line

See SC Assembly > Create Leader Distribution Line (page 501)

Attach to Viewport Tracking

See SC Assembly > Attach to Viewport Tracking (page 501)

Redistribute Leaders

See SC Assembly > Redistribute Leaders (page 502)

Set Leader Insertion Mode

See SC Assembly > Set Leader Insertion Mode (page 501)

Adjust Leader Spacing

See SC Assembly > Adjust Leader Spacing (page 500)

Transfer Leaders to Other Line

See SC Assembly > Transfer Leaders to Other Line (page 502)

Nest Toolbar

Insert Parts

See SC Plate Nest > Part > Insert (page 514)

Snap

See SC Plate Nest > Part > Snap (page 517)

Slide

See SC Plate Nest > Part > Slide Along (page 515)

Align

See SC Plate Nest > Part > Align (page 516)

Move & Rotate

See SC Plate Nest > Part > Move/Rotate (page 515)

Nest Layers

See SC Plate Nest > Layers > Visibility (page 518)

Create Bridge

See SC Plate Nest > Bridge > New (page 525)

Profile Nesting

See ShipConstructor > Profile Nesting (page 386)

Ribbon Tabs

Nest and Profile Plots Tab

Plate Nest Panel

Insert Part

See SC Plate Nest > Part > Insert (page 514)

Re-Insert Part

See SC Plate Nest > Part > Re-Insert (page 515)

Remove

See SC Plate Nest > Part > Remove (page 515)

Slide Along

See SC Plate Nest > Part > Slide Along (page 515)

Move/Rotate

See SC Plate Nest > Part > Move/Rotate (page 515)

Align

See SC Plate Nest > Part > Align (page 516)

Rotate Shortest Dimension

See SC Plate Nest > Part > Rotate Shortest Dimension (page 516)

List

See SC Plate Nest > Part > List (page 517)

Snap

See SC Plate Nest > Part > Snap (page 517)

Align with H or V

See SC Plate Nest > Part > Align with H or V (page 517)

Plot

See SC Plate Nest > Nest > Plot (page 513)

Nest Manager

See SC Plate Nest > Nest Manager (page 520)

New

See SC Plate Nest > Nest > New (page 511)

Delete

See SC Plate Nest > Nest > Delete (page 512)

Find

See SC Plate Nest > Nest > Find (page 512)

Re-Nest

See SC Plate Nest > Nest > Re-Nest (page 512)

Edit Stock Plate

See SC Plate Nest > Nest > Edit Stock Plate (page 513)

Check Nests and BOM

See SC Plate Nest > Check Nests and BOM (page 521)

Update Nests and BOM

See SC Plate Nest > Update Nests and BOM (page 522)

Nest Collision Check

See SC Plate Nest > Nest Collision Check (page 522)

List Un-nested Parts

See SC Plate Nest > Part > List Un-nested Parts (page 518)

New

See SC Plate Nest > Remnant > New (page 513)

Delete

See SC Plate Nest > Remnant > Delete (page 514)

Edit

See SC Plate Nest > Remnant > Edit Remnant (page 514)

Edit Label

See SC Plate Nest > Remnant > Edit Remnant Label (page 514)

Insert Keywords

See SC Plate Nest > Insert Keywords (page 519)

Auto Label from BOM Index

See SC Plate Nest > Auto Label from BOM Index (page 519)

Label Properties

See SC Plate Nest > Label Properties (page 520)

Visibility

See SC Plate Nest > Layers > Visibility (page 518)

Activate All

See SC Plate Nest > Layers > Activate All (page 519)

Flip NC-Pyros

See SC Plate Nest > Layers > Flip NC-Pyros (page 519)

Assign to NC-Machine

See SC Plate Nest > Nest > Assign to NC-Machine (page 513)

New

See SC Plate Nest > Bridge > New (page 525)

Update

See SC Plate Nest > Bridge > Update (page 526)

Delete

See SC Plate Nest > Bridge > Delete (page 525)

Import From

See SC Plate Nest > NC-Pyros > Import From (page 525)

Export To

See SC Plate Nest > NC-Pyros > Export To (page 523)

Export Nest Parts to DWG

See SC Plate Nest > Part > Export Nest Parts to DWG (page 518)

Export Nest Parts to DXF

See SC Plate Nest > Part > Export Nest Parts to DXF (page 518)

Profile Nest Panel

Profile Nesting

See ShipConstructor > Profile Nesting (page 386)

Profile Plots Panel

Insert Profile Plots

See SC Profile Plots > Insert Profile Plots (page 529)

Options

See SC Profile Plots > Options (page 532)

Zoom

See SC Profile Plots > Zoom (page 536)

Check Profile Drawings

See SC Profile Plots > Check Profile Drawings (page 538)

Update All BOMs

See SC Profile Plots > Update All BOMs (page 539)

Print Plots

See SC Profile Plots > Print Plots (page 537)

Plate Nest Template Panel

Insert Keywords

See SC Plate Nest Template > Insert Keywords (page 526)

Define Border

See SC Plate Nest Template > Define Border (page 526)

Check Nest Template Drawing

See SC Plate Nest Template > Check Drawing (page 529)

Update from a Previous Version

See SC Plate Nest Template > Update from a Previous Version (page 529)

Profile Plot Template Panel

Insert Keywords

See SC Profile Plot Template > Insert Keywords (page 540)

Replace Legacy Keywords

See SC Profile Plot Template > Replace Legacy Keywords (page 543)

Draw Border

See SC Profile Plot Template > Draw Border (page 539)

Draw Web Area View

See SC Profile Plot Template > Draw Web View Area (page 540)

Profile Sheet Template Panel

Insert Keywords

See SC Profile Sheet Template > Insert Keywords (page 544)

Replace Legacy Keywords

See SC Profile Sheet Template > Replace Legacy Keywords (page 544)

Draw Border

See SC Profile Sheet Template > Draw Border (page 543)

Draw Area

See SC Profile Sheet Template > Draw Area (page 543)

Structure Production Tab

Modeling/Production Utilities Panel

Global Dimension to Point

See SC Assembly > Global Dimension to Point (page 503)

Check Local Interferences

See SC Utilities > Check Local Interferences (page 466)

Clip Current View

See SC Utilities > Clip Current View (page 455)

Remove Clip

See SC Utilities > Remove Clip (page 456)

List Item within Block/Xref

See SC Utilities > List Item within Block/Xref (page 466)

Dihedral Angle

See SC Utilities > Dihedral Angle (page 461)

Create Quality Matrix

See SC Utilities > Create Quality Matrix (page 462)

Flip UCS X

See SC Utilities > UCS > Flip UCS X (page 453)

Flip UCS Y

See SC Utilities > UCS > Flip UCS Y (page 453)

Swap UCS XY Axis

See SC Utilities > UCS > Swap UCS XY Axis (page 454)

Export to DWG

See SC Utilities > Export > Export to DWG (page 466)

Export to NWC

See SC Utilities > Export > Export to NWC (page 467)

Export to NWD

See SC Utilities > Export > Export to NWD (page 467)

Labeling Panel

Relabel All

See SC Assembly > Label > Relabel All (page 496)

Relabel from BOM

See SC Assembly > Label > Relabel from BOM (page 497)

Relabel from Parts

See SC Assembly > Label > Relabel from Parts (page 497)

Manual Label from BOM

See SC Assembly > Label > Manual Label from BOM (page 496)

Copy Label from BOM

See SC Assembly > Label > Copy Label from BOM (page 496)

Label on Visible Edge On/Off

See SC Assembly > Label > Label on Visible Edge On/Off (page 497)

Curved Plates in Visible Edge Detection On/Off

See SC Assembly > Label > Curved Plates in Visible Edge Detection On/Off (page 497)

Adjacent Part Edge Tolerance

See SC Assembly > Label > Adjacent Part Edge Tolerance (page 498)

Edge Length Filter

See SC Assembly > Label > Edge Length Filter (page 498)

Edge Determination Minimum Angle

See SC Assembly > Label > Edge Determination Minimum Angle (page 498)

Label Reset Automatic Settings

See SC Assembly > Label > Label Reset Automatic Settings (page 498)

BOM Panel

Update BOMs

See SC Assembly > Update BOMs (page 495)

Edit Collector Options

See SC Assembly > Edit Collector Options (page 495)

Assembly Drawing Panel

Update Drawing

See SC Assembly > Update Drawing (page 490)

Update All Keywords

See SC Assembly > Update All Keywords (page 492)

CG Point

See SC Assembly > CG Point (page 491)

Approval Drawing Panel

Insert Group

See SC Approval > Insert Group (page 488)

Save As Bound Approval

See SC Approval > Save As Bound Approval (page 489)

Production Panel

Insert BOM Table

See SC Assembly > Insert BOM Table (page 495)

Edit Collector Options

See SC Assembly > Edit Collector Options (page 495)

Toggle List Only Visible

See SC Assembly > Toggle List Only Visible (page 495)

Insert Keywords

See SC Assembly > Insert Keywords (page 491)

Orientation Icon

See SC Assembly > Orientation Icon (page 490)

Update from a Previous Version

See SC Assembly Template > Update from a Previous Version (page 508)

Distribution Line Panel

Create Leader Distribution Line

See SC Assembly > Create Leader Distribution Line (page 501)

Attach to Viewport Tracking

See SC Assembly > Attach to Viewport Tracking (page 501)

Redistribute Leaders

See SC Assembly > Redistribute Leaders (page 502)

Set Leader Insertion Mode

See SC Assembly > Set Leader Insertion Mode (page 501)

Adjust Leader Spacing

See SC Assembly > Adjust Leader Spacing (page 500)

Transfer Leaders to Other Line

See SC Assembly > Transfer Leaders to Other Line (page 502)

Property Label Panel

Property Label

See SC Utilities > Property Labels > Property Label (page 465)

Copy Field Label

See SC Utilities > Property Labels > Copy Field Label (page 465)

Copy Field Label Quick

See SC Utilities > Property Labels > Copy Field Label Quick (page 465)

Edit Field Label

See SC Utilities > Property Labels > Edit Field Label (page 465)

Replace Object References

See SC Utilities > Property Labels > Replace Object References (page 465)

Set Single Click Field Label

See SC Utilities > Property Labels > Set Single Click Field Label (page 466)

Distributed Systems Production Tab

Modeling/Production Utilities Panel

Global Dimension to Point

See SC Assembly > Global Dimension to Point (page 503)

Check Local Interferences

See SC Utilities > Check Local Interferences (page 466)

Clip Current View

See SC Utilities > Clip Current View (page 455)

Remove Clip

See SC Utilities > Remove Clip (page 456)

List Item within Block/Xref

See SC Utilities > List Item within Block/Xref (page 466)

Dihedral Angle

See SC Utilities > Dihedral Angle (page 461)

Create Quality Matrix

See SC Utilities > Create Quality Matrix (page 462)

Flip UCS X

See SC Utilities > UCS > Flip UCS X (page 453)

Flip UCS Y

See SC Utilities > UCS > Flip UCS Y (page 453)

Swap UCS XY Axis

See SC Utilities > UCS > Swap UCS XY Axis (page 454)

Export to DWG

See SC Utilities > Export > Export to DWG (page 466)

Export to NWC

See SC Utilities > Export > Export to NWC (page 467)

Export to NWD

See SC Utilities > Export > Export to NWD (page 467)

Labeling Panel

Relabel All

See SC Assembly > Label > Relabel All (page 496)

Relabel from BOM

See SC Assembly > Label > Relabel from BOM (page 497)

Relabel from Parts

See SC Assembly > Label > Relabel from Parts (page 497)

Manual Label from BOM

See SC Assembly > Label > Manual Label from BOM (page 496)

Copy Label from BOM

See SC Assembly > Label > Copy Label from BOM (page 496)

Label on Visible Edge On/Off

See SC Assembly > Label > Label on Visible Edge On/Off (page 497)

Curved Plates in Visible Edge Detection On/Off

See SC Assembly > Label > Curved Plates in Visible Edge Detection On/Off (page 497)

Adjacent Part Edge Tolerance

See SC Assembly > Label > Adjacent Part Edge Tolerance (page 498)

Edge Length Filter

See SC Assembly > Label > Edge Length Filter (page 498)

Edge Determination Minimum Angle

See SC Assembly > Label > Edge Determination Minimum Angle (page 498)

Label Reset Automatic Settings

See SC Assembly > Label > Label Reset Automatic Settings (page 498)

BOM Panel

Update BOMs

See SC Assembly > Update BOMs (page 495)

Edit Collector Options

See SC Assembly > Edit Collector Options (page 495)

Spool Drawing Panel

Update Drawing

See SC Spool Drawing > Update Drawing (in Pipe of HVAC Manual)

Rotate Dimension

See SC Spool Drawing > Rotate Dimension (in Pipe of HVAC Manual)

Align Dimension Text to Current View

See SC Spool Drawing > Align Dimension Text to Current View (in Pipe of HVAC Manual)

Re- Dimension

See SC Spool Drawing > Re-Dimension (in Pipe of HVAC Manual)

Update All Keywords

See SC Spool Drawing > Update All Keywords (in Pipe of HVAC Manual)

Label Connecting Spool/Assembly

See SC Spool Drawing > Label Connecting Spool/Assembly (in Pipe of HVAC Manual)

CG Point

See SC Spool Drawing > CG Point (in Pipe of HVAC Manual)

Arrangement Drawing Panel

Update Drawing

See SC Arrangement > Update Drawing (in Pipe of HVAC Manual)

Update All Keywords

See SC Arrangement > Update All Keywords (in Pipe of HVAC Manual)

Spool Template Panel

Drawing Options

See SC Spool Template > Drawing Options (in Pipe of HVAC Manual)

Production Panel

Insert BOM Table

See SC Assembly > Insert BOM Table (page 495)

Edit Collector Options

See SC Assembly > Edit Collector Options (page 495)

Toggle List Only Visible

See SC Assembly > Toggle List Only Visible (page 495)

Insert Keywords

See SC Assembly > Insert Keywords (page 491)

Orientation Icon

See SC Assembly > Orientation Icon (page 490)

Update from a Previous Version

See SC Assembly Template > Update from a Previous Version (page 508)

Distribution Line Panel

Create Leader Distribution Line

See SC Assembly > Create Leader Distribution Line (page 501)

Attach to Viewport Tracking

See SC Assembly > Attach to Viewport Tracking (page 501)

Redistribute Leaders

See SC Assembly > Redistribute Leaders (page 502)

Set Leader Insertion Mode

See SC Assembly > Set Leader Insertion Mode (page 501)

Adjust Leader Spacing

See SC Assembly > Adjust Leader Spacing (page 500)

Transfer Leaders to Other Line

See SC Assembly > Transfer Leaders to Other Line (page 502)

Property Label Panel

Property Label

See SC Utilities > Property Labels > Property Label (page 465)

Copy Field Label

See SC Utilities > Property Labels > Copy Field Label (page 465)

Copy Field Label Quick

See SC Utilities > Property Labels > Copy Field Label Quick (page 465)

Edit Field Label

See SC Utilities > Property Labels > Edit Field Label (page 465)

Replace Object References

See SC Utilities > Property Labels > Replace Object References (page 465)

Set Single Click Field Label

See SC Utilities > Property Labels > Set Single Click Field Label (page 466)

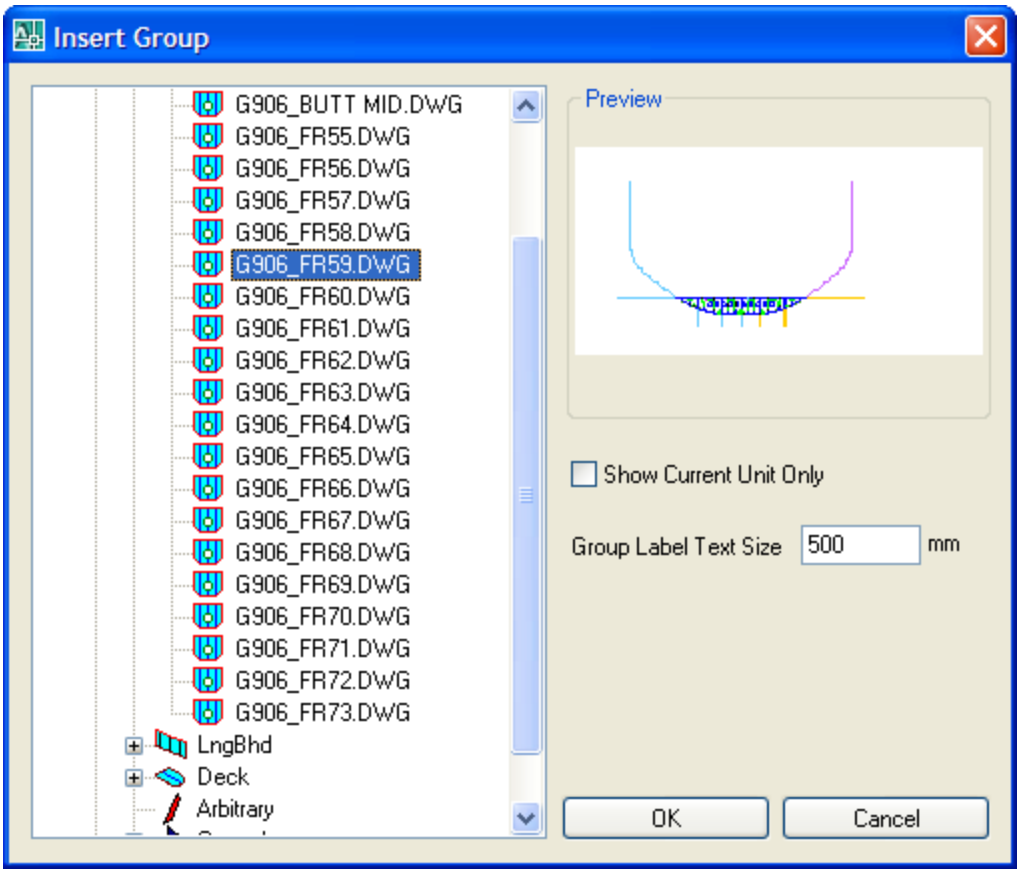
SC Approval Menu

The SC Approval menu appears in approval drawings (see Approval Drawings (page 215)).

SC Approval > Insert Group

Button .....
Ribbon ..... Structure Production tab > Approval Drawing panel > Insert Group
Menu ..... SC Approval > Insert Group
Toolbar..... None
Command ..... SCINSERTAPPROVAL
Permissions ... General > Edit Approval
Procedure..... Insert a Planar Group into an Approval Drawing (page 216)

Inserts a planar group into an approval drawing.



Insert Planar group options

Show Current Unit Only – Limits the list of available drawings to insert to the current unit to make it faster to show and easier to select from a smaller list.

Group Label Text Size – Each inserted group has a label of the group name as TEXT below the geometry. This value specifies the height of that text. The text style is Standard.

SC Approval > Save As Bound Approval

Button .....
Ribbon ..... Structure Production tab > Approval Drawing panel > Save As Bound Approval
Menu ..... SC Approval > Save As Bound Approval
Toolbar..... None
Command ..... SCBINDAPPROVAL
Permissions ... General > Edit Approval
Procedure..... Save a Bound Approval Drawing (page 217)

Saves an approval drawing as a bound approval drawing.

SC Assembly Menu

The SC Assembly menu is available within assembly drawings. See Assembly Drawings (page 218).

SC Assembly > Update Drawing

Button .....
Ribbon ..... Structure Production tab > Assembly Drawing panel > Update Drawing
Menu ..... SC Assembly > Update Drawing
Toolbar..... None
Command ..... SCUPDATEDWG
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Update Parts in an Assembly Drawing (page 229)

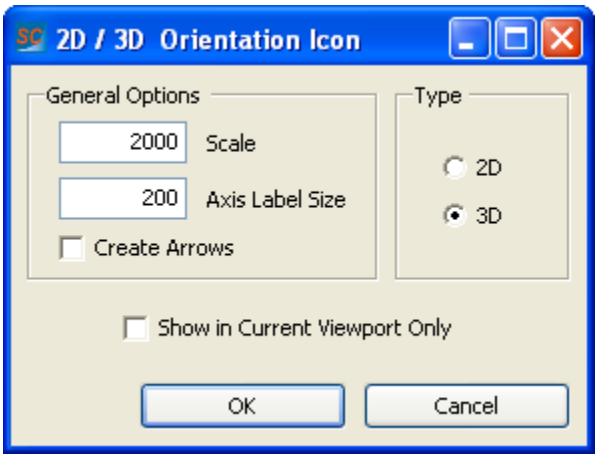
Updates the parts and keywords within an assembly drawing. Annotations are not updated.

SC Assembly > Orientation Icon

Button .....
Ribbon ..... Structure Production tab > Modeling/Production Utilities panel > Orientation Icon
Menu ..... SC Assembly > Orientation Icon
Toolbar..... None
Command ..... SCASSORIENT
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Insert an Orientation Icon into an Assembly Drawing (page 236)

Inserts an orientation icon into an assembly drawing. The orientation icon can be 3D for iso drawings or 2D for 2D drawings. The style of the icon is not changeable currently.

3D/2D Orientation Icon Window

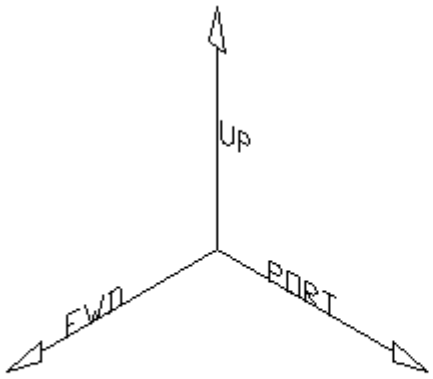


Scale – The size of the icon in project units.

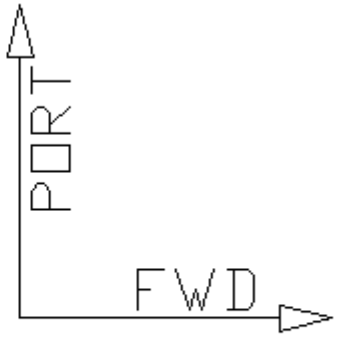
Axis Label Size – The size of the direction text.

Create Arrows – Decides if there are arrows at the end of the axis lines.

Type 3D, 2D



3D



2D

Show in current viewport only – If checked, will place the icon on a layer that is only visible in the current viewport.

SC Assembly > CG Point

Button .....
Ribbon ..... Structure Production tab > Assembly Drawing panel > CG Point
Menu ..... SC Assembly > CG Point
Toolbar..... None
Command ..... SCINSERTCG
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Mark an Assembly's CG Position in an Assembly Drawing (page 236)

The command inserts a SConCGPoint object at a center of gravity (CG) position of the current assembly. The SConCGPoint is exactly like an AutoCAD point object except you cannot move it. The command also changes the current Point style to ⊕ (34). You can change the style of the CG point by using the point style options. The orientation of the point is based on the orientation of the current UCS. If a SConCGPoint already exists in drawing, it will be moved to the correct CG position.

Note: Running SC Assembly > Update Drawing (page 490) will move the SConCGPoint to the updated CG position.

SC Assembly > Insert Keywords

Button .....
Ribbon ..... Structure Production tab > Modeling/Production Utilities panel > Insert Keywords
Menu ..... SC Assembly > Insert Keywords
Toolbar..... None
Command ..... SCINSERTKEYWORD
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Insert Keywords into an Assembly Drawing (page 231)

Inserts keywords into an assembly drawing. See also SC Assembly Template > Insert Keywords (page 505).

SC Assembly > Update All Keywords

Button .....
Ribbon ..... Structure Production tab > Assembly Drawing panel > Update All Keywords
Menu ..... SC Assembly > Update All Keywords
Toolbar..... None
Command ..... SCUPDATEALLKEYWORDS
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Update All Keywords (page 231)

Updates all keywords in paper space in the drawing.

SC Assembly > Structure Drawing Options

Button .....
Ribbon ..... Structure Production tab > Drawing Options panel > Structure Drawing Options
Menu ..... SC Assembly > Structure Drawing Options
Toolbar..... Structure Display >
Command ..... SCSTRUCTDISPLAY
Permissions ... None
Procedure..... SC Structure > Show / Hide Options

Displays the structure drawing options.

SC Assembly > Pipe Drawing Options

Button .....
Ribbon ..... Structure Production tab > Drawing Options panel > Pipe Drawing Options
Menu ..... SC Assembly > Pipe Drawing Options
Toolbar..... Pipe >
Command ..... SCDWGOPTIONSPIPE
Permissions ... None
Procedure..... See the Pipe manual for details

Displays the pipe drawing options.

SC Assembly > HVAC Drawing Options

Button .....
Ribbon ..... Structure Production tab > Drawing Options panel > HVAC Drawing Options
Menu ..... SC Assembly > HVAC Drawing Options
Toolbar..... HVAC >
Command ..... SCDWGOPTIONSHVAC
Permissions ... None
Procedure..... See the HVAC manual for details.

Displays the HVAC drawing options.

SC Assembly > Equipment Drawing Options

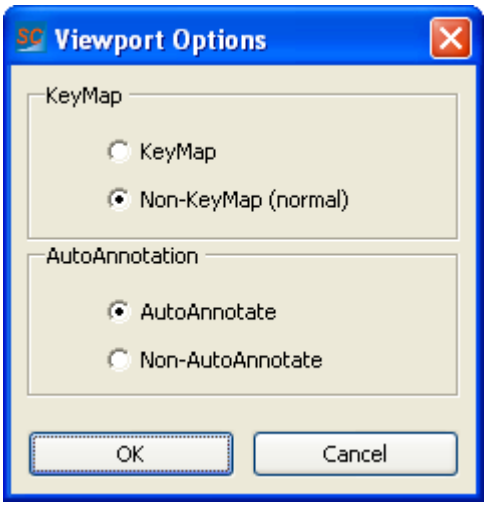
Button .....
Ribbon ..... Structure Production tab > Drawing Options panel > Equipment Drawing Options
Menu ..... SC Assembly > Equipment Drawing Options
Toolbar..... Equipment >
Command ..... SCDWGOPTIONSEQUIP
Permissions ... None
Procedure..... See the Equipment manual for details.

Displays the Equipment drawing options.

SC Assembly > Viewport Options

Button .....
Ribbon ..... Structure Production tab > Drawing Options panel > Viewport Options
Menu ..... SC Assembly > Viewport Options
Toolbar..... None
Command ..... SCVPORTOPTIONS
Permissions ... None

Lets you indicate whether an assembly drawing viewport is a keymap viewport or a normal viewport and whether ShipConstructor will insert automatic annotations into the viewport.



SC Assembly > Set Viewport Display Options

Command ..... SCVIEWPORTDISPOPTIONS
Permissions ... None

This command is no longer supported. SCDWGOPTIONSVIEWPORT should be used instead.

Button .....
Ribbon ..... Structure Production tab > Drawing Options panel > Set Viewport Display Options
Menu ..... SC Assembly > Set Viewport Display Options
Toolbar..... None
Command ..... SCDWGOPTIONSVIEWPORT
Permissions ... None
Procedure..... Set Up Assembly Drawing Template Viewports (page 220)

Configure the viewport specific display options for a specific viewport.

SC Assembly > Save Drawing Options

Button .....
Ribbon ..... Production tab > Drawing Options panel > Save Drawing Options
Menu ..... Assembly > Save Drawing Options
Toolbar..... None
Command ..... SCSAVEDRAWINGOPTIONSTOFILE
Permissions ... None
Procedure..... None

Saves the current drawing's drawing options to a file. An SCDO file is created and it can be used either the drawing options or the viewport display options.

SC Assembly > Load Drawing Options

Button .....
Ribbon ..... Production tab > Drawing Options panel > Load Drawing Options
Menu ..... Assembly > Load Drawing Options
Toolbar..... None
Command ..... SCLOADDRAWINGOPTIONSFROMFILE
Permissions ... None
Procedure..... None

Loads drawing options from a selected SCDO file into the current drawing.

SC Assembly > Save Viewport Display Options

Button .....
Ribbon ..... Production tab > Drawing Options panel > Save Viewport Display Options
Menu ..... Assembly > Save Viewport Display Options
Toolbar..... None
Command ..... SCSAVEVIEWPORTDRAWINGOPTIONSTOFILE
Permissions ... None
Procedure..... None

Saves the selected viewport's display options to a file. An SCDO file is created and it can be used either the drawing options or the viewport display options.

SC Assembly > Load Viewport Display Options

Button .....
Ribbon ..... Production tab > Drawing Options panel > Load Viewport Display Options
Menu ..... Assembly > Load Viewport Display Options
Toolbar..... None
Command ..... SCLOADVIEWPORTDRAWINGOPTIONSFROMFILE
Permissions ... None
Procedure..... None

Loads display options from a selected SCDO file and applies them to the selected viewport.

SC Assembly > Insert BOM Table

Button .....
Ribbon ..... Structure Production tab > BOM panel > Insert BOM Table
Menu ..... SC Assembly > Insert BOM Table
Toolbar..... None
Command ..... SCINSERTBOM
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Insert a BOM Table into an Assembly Drawing Template (page 222)

Inserts a BOM table.

SC Assembly > Update BOMs

Button .....
Ribbon ..... Structure Production tab > BOM panel > Update BOMs
Menu ..... SC Assembly > Update BOMs
Toolbar..... None
Command ..... SCUPDATEBOMS
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Update BOMs (page 233)

Updates the BOM Table.

SC Assembly > Edit Collector Options

Button .....
Ribbon ..... Structure Production tab > BOM panel > Edit Collector Options
Menu ..... SC Assembly > Edit Collector Options
Toolbar..... None
Command ..... SCTWEAKBOMOPTIONS
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Edit BOM Collector Options (page 233)

Allows the user to change the Pipe/HVAC BOM Collector options.

SC Assembly > Toggle List Only Visible

Button .....
Ribbon ..... Structure Production tab > BOM panel > Toggle List Only Visible
Menu ..... SC Assembly > Toggle List Only Visible
Toolbar..... None
Command ..... SCTOGGLELISTVISIBLE
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... List Only Visible (page 235)

Toggles the variable in the BOM to list only those parts visible or not.

SC Assembly > Label > Manual Label from BOM

Button .....
Ribbon ..... Structure Production tab > Labeling panel > Manual Label from BOM
Menu ..... SC Assembly > Label > Manual Label from BOM
Toolbar..... Assembly >
Command ..... SCLABELMANUAL
Permissions ... None
Procedure..... Manual Labeling from BOM (page 238)

Manaully label parts from a BOM. This is an alternative method to automatic labeling. The label style and text contents are used from the selected BOM. The selection of the leader arrow is the pickpoint where you selected the part. The number of leader segments follows the label style setting.

This command can be used in conjunction with Copy Label from BOM so complete the labeling. If you select a part that has been previously labeled then the previous label is removed.

SC Assembly > Label > Copy Label from BOM

Button .....
Ribbon ..... Structure Production tab > Labeling panel > Copy Label from BOM
Menu ..... SC Assembly > Label > Copy Label from BOM
Toolbar..... Assembly >
Command ..... SCLABELCOPY
Permissions ... None
Procedure..... Copy Label from BOM (page 246)

Lets you create labels that are copies of an existing label with the contents changed to reflect the first BOM column of the selected part. The copied label will be created with the label text and styles from the BOM table and geometry information from the source label.

SC Assembly > Label > Relabel All

Button .....
Ribbon ..... Structure Production tab > Labeling panel > Relabel All
Menu ..... SC Assembly > Label > Relabel All
Toolbar..... None
Command ..... SCRELABELALL
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Relabel All (page 246)

Generates labels for all new BOM items and places them on the _ANNOTATION layer.

Labels are generated only once. Subsequent commands only update the label text to reflect the Bill of Materials.

Even though labels are placed in Paper Space, they now track the objects in Model Space so that if the user pans, zooms, or orbits in the viewport, the labels track accordingly and remain synchronized to the objects that they represent.

SC Assembly > Label > Relabel from BOM

Button .....
Ribbon ..... Structure Production tab > Labeling panel > Relabel from BOM
Menu ..... SC Assembly > Label > Relabel from BOM
Toolbar..... None
Command ..... SCRELABELFROMBOM
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Relabel from BOM (page 246)

Relabels parts from a selected BOM and places them on the _ANNOTATION layer.

Labels are generated only once. Subsequent commands only update the label text to reflect the Bill of Materials.

Even though labels are placed in Paper Space, they now track the objects in Model Space so that if the user pans, zooms, or orbits in the viewport, the labels track accordingly and remain synchronized to the objects that they represent.

SC Assembly > Label > Relabel from Parts

Button .....
Ribbon ..... Structure Production tab > Labeling panel > Relabel from Parts
Menu ..... SC Assembly > Label > Relabel from Parts
Toolbar..... None
Command ..... SCRELABELFROMPARTS
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Relabel from Parts (page 246)

Relabels selected parts from the BOMs.

SC Assembly > Label > Label on Visible Edge On/Off

Button .....
Ribbon ..... Structure Production tab > Labeling panel > Label on Visible Edge On/Off
Menu ..... SC Assembly > Label > Label on Visible Edge On/Off
Toolbar..... None
Command ..... SCLABELONVISIBLEEDGE
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Label on Visible Edge On/Off (page 249)

Turns labels automatic label generation from label to piecemark to label on visible part edge.

SC Assembly > Label > Curved Plates in Visible Edge Detection On/Off

Button .....
Ribbon ..... Structure Production tab > Labeling panel > Curved Plates in Visible Edge Detection On/Off
Menu ..... SC Assembly > Label > Curved Plates in Visible Edge Detection On/Off
Toolbar..... None
Command ..... SCLABELREMOVECURVEDPLATEHLR
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Curved Plates in Visible Edge Detection On/Off (page 250)

Turns on option to include curved plates in hidden line calculation when label on visible part edge is on.

SC Assembly > Label > Adjacent Part Edge Tolerance

Button .....
Ribbon ..... Structure Production tab > Labeling panel > Adjacent Part Edge Tolerance
Menu ..... SC Assembly > Label > Adjacent Part Edge Tolerance
Toolbar..... None
Command ..... SCLABELADJACENTTOL
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Adjacent Part Edge Tolerance (page 249)

Edges will be less likely to be labeled if they are closer than this distance to an edge of another part.

SC Assembly > Label > Edge Length Filter

Button .....
Ribbon ..... Structure Production tab > Labeling panel > Edge Length Filter
Menu ..... SC Assembly > Label > Edge Length Filter
Toolbar..... None
Command ..... SCLABELEDGELENGTHFILTER
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Edge Length Filter (page 250)

Edges shorter than this percentage will be filtered out from consideration of being pointed to by the labe.

SC Assembly > Label > Edge Determination Minimum Angle

Button .....
Ribbon ..... Structure Production tab > Labeling panel > Edge Determination Minimum Angle
Menu ..... SC Assembly > Label > Edge Determination Minimum Angle
Toolbar..... None
Command ..... SCLABELEDGEDETERMINATIONANGLE
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Edge Determination Minimum Angle (page 251)

Line segments will be considered part of the same edge if they are less than this angle away from tangent.

SC Assembly > Label > Label Reset Automatic Settings

Button .....
Ribbon ..... Structure Production tab > Labeling panel > Label Reset Automatic Settings
Menu ..... SC Assembly > Label > Label Reset Automatic Settings
Toolbar..... None
Command ..... SCLABELRESETALL
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Label Reset Automatic Settings (page 251)

Resets the automatic label generation settings in the current drawing to use the project settings.

SC Assembly > BOM Revisions > New BOM Revision in Current Layout

Button .....
Ribbon ..... Production tab > BOM Revisions panel > New BOM Revision in Current Layout
Menu ..... SC Assembly > BOM Revisions > New BOM Revision in Current Layout
Toolbar..... None
Command ..... SCPRODREVNEW
Permissions ... Structure > Assembly Drawing Revisions – Add/Edit
Procedure..... New Revision (page 253)

Creates a revision in the current layout.

SC Assembly > BOM Revisions > New BOM Revision in All Layouts

Button .....
Ribbon ..... Production tab > BOM Revisions panel > New BOM Revision in All Layouts
Menu ..... SC Assembly > BOM Revisions > New BOM Revision in All Layouts
Toolbar..... None
Command ..... SCPRODREVNEWALL
Permissions ... Structure > Assembly Drawing Revisions – Add/Edit
Procedure..... New Revision (page 253)

Creates a revision in all layouts.

SC Assembly > BOM Revisions > Delete BOM Revision in Current Layout

Button .....
Ribbon ..... Production tab > BOM Revisions panel > Delete BOM Revision in Current Layout
Menu ..... SC Assembly > BOM Revisions > Delete BOM Revision in Current Layout
Toolbar..... None
Command ..... SCPRODREVDEL
Permissions ... Structure > Assembly Drawing Revisions – Remove
Procedure..... Delete Revision (page 253)

Deletes the current revision in the current layout.

SC Assembly > BOM Revisions > Delete BOM Revision in All Layouts

Button .....
Ribbon ..... Production tab > BOM Revisions panel > Delete BOM Revision in All Layouts
Menu ..... SC Assembly > BOM Revisions > Delete BOM Revision in All Layouts
Toolbar..... None
Command ..... SCPRODREVDELALL
Permissions ... Structure > Assembly Drawing Revisions – Remove
Procedure..... Delete Revision (page 253)

Deletes the current revision in all layouts.

SC Assembly > BOM Revisions > Delete All BOM Revisions in Current Layout

Button .....
Ribbon ..... Production tab > BOM Revisions panel > Delete All BOM Revision in Current Layout
Menu ..... SC Assembly > BOM Revisions > Delete All BOM Revision in Current Layout
Toolbar..... None
Command ..... SCPRODREVCLEAR
Permissions ... Structure > Assembly Drawing Revisions – Remove
Procedure..... Delete All Revisions (page 253)

Deletes all revisions in the current layout.

SC Assembly > BOM Revisions > Delete All BOM Revisions in All Layouts

Button .....
Ribbon ..... Production tab > BOM Revisions panel > Delete All BOM Revision in All Layouts
Menu ..... SC Assembly > BOM Revisions > Delete All BOM Revision in All Layouts
Toolbar..... None
Command ..... SCPRODREVCLEARALL
Permissions ... Structure > Assembly Drawing Revisions – Remove
Procedure..... Delete All Revisions (page 253)

Deletes all revisions in all layouts.

SC Assembly > BOM Revisions > List BOM Revisions

Button .....
Ribbon ..... Production tab > BOM Revisions panel > List BOM Revisions
Menu ..... SC Assembly > BOM Revisions > List BOM Revisions
Toolbar..... None
Command ..... SCPRODREVLIST
Permissions ... None
Procedure..... List Revisions (page 253)

List the information about the revisions in the current layout.

SC Assembly > Adjust Leader Spacing

Button .....
Ribbon ..... Structure Production tab > Distribution Line panel > Adjust Leader Spacing
Menu ..... SC Assembly > Adjust Leader Spacing
Toolbar..... Leader Utils
Command ..... SCCREATEDISTRIBUTIONLINE
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Adjust Leader Spacing

Adjust minmum leader spacing setting on the leader distribution line. This setting will be used when Redistribute Leaders is used.

SC Assembly > Attach to Viewport Tracking

Button .....
Ribbon ..... Structure Production tab > Distribution Line panel > Attach to Viewport Tracking
Menu ..... SC Assembly > Attach to Viewport Tracking
Toolbar..... Leader Utils
Command ..... SCATTDL
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Attach to Viewport Tracking

Viewport tracking allows certain objects to translate in accordance with pan and zoom operations in the viewport. The following objects can be attached to viewport tracking:

- Distribution lines
- ShipConstructor labels (SConLabel)
- AutoCAD leaders
- MText
- AutoCAD Text
- Lines

SC Assembly > Create Leader Distribution Line

Button .....
Ribbon ..... Structure Production tab > Distribution Line panel > Create Leader Distribution Line
Menu ..... SC Assembly > Create Leader Distribution Line
Toolbar..... Leader Utils
Command ..... SCCREATEDISTRIBUTIONLINE
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Create Leader Distribution Line

Creates a line that can be used to easily align and move labels as a group. This leader distribution line is a non-printed line.

SC Assembly > Set Leader Insertion Mode

Button .....
Ribbon ..... Structure Production tab > Distribution Line panel > Set Leader Insertion Mode
Menu ..... SC Assembly > Set Leader Insertion Mode
Toolbar..... Leader Utils
Command ..... SCDISTLINESETINSERTIONMODE
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Set Leader Insertion Mode (page 321)

Adjusts the insertion mode of all labels on line. The four possible modes are: Left, Right, Center, closest Horizontal. The closest horizontal mode will use the left or right alignment depending on which one is closer to the leader point.

SC Assembly > Redistribute Leaders

Button .....
Ribbon ..... Structure Production tab > Distribution Line panel > Redistribute Leaders
Menu ..... SC Assembly > Redistribute Leaders
Toolbar..... Leader Utils
Command ..... SCLABELREDISTRIBUTE
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Redistribute Leaders (page 321)

Redistribute leaders on the leader distribution line in one of the three distribution modes – Nearest, Center and Equidistant. All leaders will be automatically uncrossed and distributed on the line.

SC Assembly > Transfer Leaders to Other Line

Button .....
Ribbon ..... Structure Production tab > Distribution Line panel > Transfer Leaders to Other Line
Menu ..... SC Assembly > Transfer Leaders to Other Line
Toolbar..... Leader Utils
Command ..... SCTRANSFERLEADERS
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Transfer Leaders to Other Line

Move leaders from one distribution line to another.

SC Assembly > Distribution line merge distance tolerance

Button .....
Ribbon ..... Structure Production tab > Distribution Line panel > Distribution line merge distance tolerance
Menu ..... SC Assembly > Distribution line merge distance tolerance
Toolbar..... None
Command ..... SCLEADERDISTLINEMAXDISTANCEBEFOREMERGE
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... None

Changes the distance tolerance used to determine if two distribution lines should be merged together. If the distribution lines are less than this distance apart and are parallel, they will be merged into one. Used in conjunction with Merge Distribution Line Angle.

SC Assembly > Distribution line merge angle tolerance

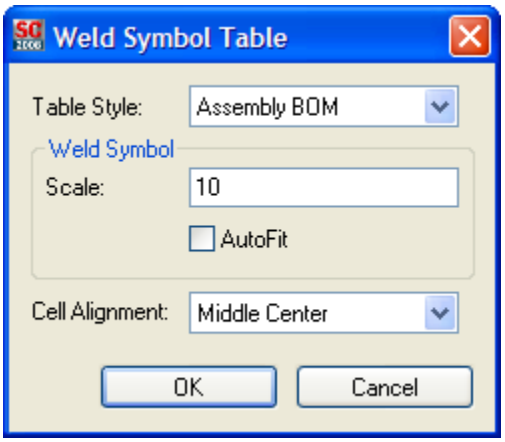
Button .....
Ribbon ..... Structure Production tab > Distribution Line panel > Distribution line merge angle tolerance
Menu ..... SC Assembly > Distribution line merge angle tolerance
Toolbar..... None
Command ..... SCLEADERDISTLINEPARALLELITYMERGEANGLE
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... None

Changes the parallelity determination angle tolerance used to determine if two distribution lines should be merged together. If the distribution lines are within the distance and the angle between the lines is less than this angle, they will be merged into one. Used in conjunction with Merge Distribution Line Distance.

SC Assembly > Insert Weld Symbol Table

Button .....
Ribbon ..... None
Menu ..... SC Assembly > Insert Weld Symbol Table
Toolbar..... None
Command ..... SCINSERTWELDSYMBOLTABLE
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Insert a Weld Symbol Table into an Assembly Drawing (page 235)

Inserts an empty 1x2 table, which will be filled with weld standard and weld scenarios for all welds in the drawing's assembly.



SC Assembly > Update Weld Symbol Tables

Button .....
Ribbon ..... None
Menu ..... SC Assembly > Update Weld Symbol Tables
Toolbar..... None
Command ..... SCUPDATEWELDSYMBOLTABLES
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Update Weld Symbol Tables in an Assembly Drawing (page 235)

Updates all weld symbol tables in the current drawing, instead of having to do a full drawing update.

SC Assembly > Global Dimension to Point

Button .....
Ribbon ..... Structure Production tab > Labeling panel > Global Dimension to Point
Menu ..... SC Assembly > Global Dimension to Point
Toolbar..... None
Command ..... SCDISTANCEFROMPOINT
Permissions ... Structure > Assembly Drawings - Edit
Procedure..... Insert a Label or Dimension From a Point to a Structural Group (page 205)

Indicates the distance from a given point to a selected plane as a label or a dimension.

SC Assembly Template Menu

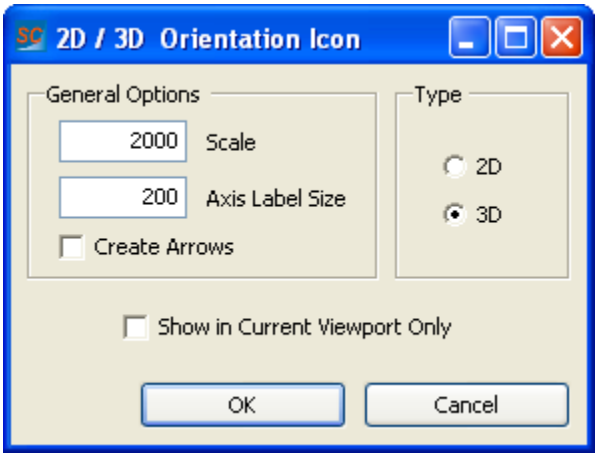
The SC Assembly Template menu appears in assembly template drawings (see Set Up Assembly Drawings (page 219)).

SC Assembly Template > Orientation Icon

Button .....
Ribbon ..... Structure Production tab > Modeling/Production Utilities panel > Orientation Icon
Menu ..... SC Assembly Template > Orientation Icon
Toolbar..... None
Command ..... SCASSORIENT
Permissions ... General > Edit Template Drawings
Procedure..... Insert an Orientation Icon into an Assembly Drawing (page 236)

Inserts an orientation icon into an assembly drawing. The orientation icon can be 3D for iso drawings or 2D for 2D drawings. The style of the icon is not changeable currently.

3D/2D Orientation Icon Window

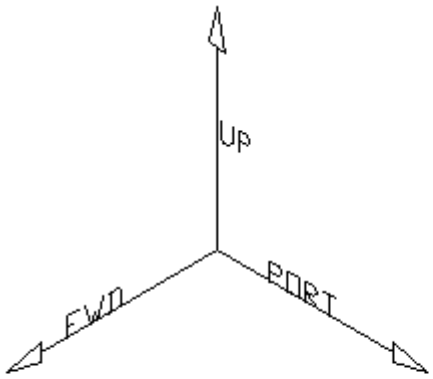


Scale – The size of the icon in project units.

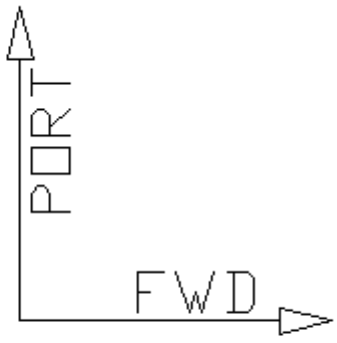
Axis Label Size – The size of the direction text.

Create Arrows – Decides if there are arrows at the end of the axis lines.

Type 3D, 2D



3D



2D

Show in current viewport only – If checked, will place the icon on a layer that is only visible in the current viewport.

SC Assembly Template > Insert Keywords

Button .....
Ribbon ..... Structure Production tab > Modeling/Production Utilities panel > Insert Keywords
Menu ..... SC Assembly Template > Insert Keywords
Toolbar..... None
Command ..... SCINSERTKEYWORD
Permissions ... General > Edit Template Drawings
Procedure..... Insert Keywords into an Assembly Drawing Template (page 221)

Lets you insert keywords into an assembly drawing template.

Keyword	Description
<ASSEMBLY CG>	The Center of Gravity of the assembly
<ASSEMBLY DESCRIPTION 1>	The Description1 User Defined Attribute of the assembly
<ASSEMBLY DESCRIPTION 2>	The Description2 User Defined Attribute of the assembly
<ASSEMBLY LCG>	Longitudinal component of the Center of Gravity of the assembly
[Build Strategy] <LEVEL : ASSEMBLY LEVEL ASSEMBLY NAME>	The level (rank) name of the assembly in every build strategy
<ASSEMBLY NAME>	The name of the assembly
<ASSEMBLY PROPERTIES>	The level name, weight, center of gravity, and finishes of the assembly
<ASSEMBLY TCG>	The transverse component of the Center of Gravity of the assembly
<ASSEMBLY VCG>	The vertical component of the Center of Gravity of the assembly
<ASSEMBLY WEIGHT>	The total weight of the assembly
<FULL ASSEMBLY NAME>	The full path of the assembly tree that this assembly belongs to
<SHORT ASSEMBLY NAME>	The name of the assembly and its direct parent
[Build Strategy] <ASSEMBLY UDA>	User Defined Attributes of the assembly in every build strategy. One keyword for each UDA

SC Assembly Template > Structure Drawing Options

Button .....
Ribbon ..... Structure Production tab > Drawing Options > Structure Drawing Options
Menu ..... SC Assembly Template > Structure Drawing Options
Toolbar..... Structure Display >
Command ..... SCSTRUCTDISPLAY
Permissions ... None
Procedure..... SC Structure > Show / Hide Options

Displays the structure drawing options.

SC Assembly Template > Pipe Drawing Options

Button .....
Ribbon ..... Structure Production tab > Drawing Options > Pipe Drawing Options
Menu ..... SC Assembly Template > Pipe Drawing Options
Toolbar..... Pipe >
Command ..... SCDWGOPTIONSPIPE
Permissions ... None
Procedure..... See the Pipe manual for details

Displays the pipe drawing options.

SC Assembly Template > HVAC Drawing Options

Button .....
Ribbon ..... Structure Production tab > Drawing Options > HVAC Drawing Options
Menu ..... SC Assembly Template > HVAC Drawing Options
Toolbar..... HVAC >
Command ..... SCDWGOPTIONSHVAC
Permissions ... None
Procedure..... See the HVAC manual for details.

Displays the HVAC drawing options.

SC Assembly Template > Equipment Drawing Options

Button .....
Ribbon ..... Structure Production tab > Drawing Options > Equipment Drawing Options
Menu ..... SC Assembly Template > Equipment Drawing Options
Toolbar..... Equipment >
Command ..... SCDWGOPTIONSEQUIP
Permissions ... None
Procedure..... See the Equipment manual for details.

Displays the Equipment drawing options.

SC Assembly Template > Viewport Options

Button .....
Ribbon ..... Structure Production tab > Drawing Options > Viewport Options

Menu ..... SC Assembly Template > Viewport Options
Toolbar..... None
Command ..... SCVPORTOPTIONS
Permissions ... None
Procedure..... Set Up Assembly Drawing Template Viewports (page 220)

Lets you indicate whether an assembly drawing template viewport is a keymap viewport or a normal viewport and whether ShipConstructor will insert automatic annotations into the viewport.

SC Assembly Template > Set Viewport Display Options

Button .....
Ribbon ..... Structure Production tab > Drawing Options > Set Viewport Display Options
Menu ..... SC Assembly Template > Set Viewport Display Options
Toolbar..... None
Command ..... SCVIEWPORTDISPOPTIONS
Permissions ... None
Procedure..... Set Up Assembly Drawing Template Viewports (page 220)

Configure the viewport specific display options for a specific viewport.

SC Assembly Template > Insert Weld Symbol Table

Button .....
Ribbon ..... None
Menu ..... SC Assembly Template > Insert Weld Symbol Table
Toolbar..... None
Command ..... SCINSERTWELDSYMBOLTABLE
Permissions ... General > Edit Template Drawings
Procedure..... Insert a Weld Symbol Table into an Assembly Drawing (page 235)

SC Assembly Template > Insert BOM Table

Button .....
Ribbon ..... Structure Production tab > BOM panel > Insert BOM Table
Menu ..... SC Assembly Template > Insert BOM Table
Toolbar..... None
Command ..... SCINSERTBOM
Permissions ... General > Edit Template Drawings
Procedure.....

SC Assembly Template > Edit Collector Options

Button .....
Ribbon ..... Structure Production tab > BOM panel > Edit Collector Options
Menu ..... SC Assembly Template > Edit Collector Options
Toolbar..... None
Command ..... SCTWEAKBOMOPTIONS
Permissions ... General > Edit Template Drawings
Procedure..... Edit BOM Collector Options (page 233)

Allows the user to change the BOM Collector options.

SC Assembly Template > Toggle List Only Visible

Button .....
Ribbon ..... Structure Production tab > BOM panel > Toggle List Only Visible
Menu ..... SC Assembly Template > Toggle List Only Visible
Toolbar..... None
Command ..... SCTOGGLELISTVISIBLE
Permissions ... General > Edit Template Drawings
Procedure..... List Only Visible (page 235)

Toggles the variable in the BOM to list only those parts visible or not.

SC Assembly Template > Update from a Previous Version

Button .....
Ribbon ..... Structure Production tab > Modeling/Production Utilities panel > Update from a Previous Version
Menu ..... SC Assembly Template > Update from a Previous Version
Toolbar..... None
Command ..... SCUPDATEPREVIOUSTEMP
Permissions ... None
Procedure.....

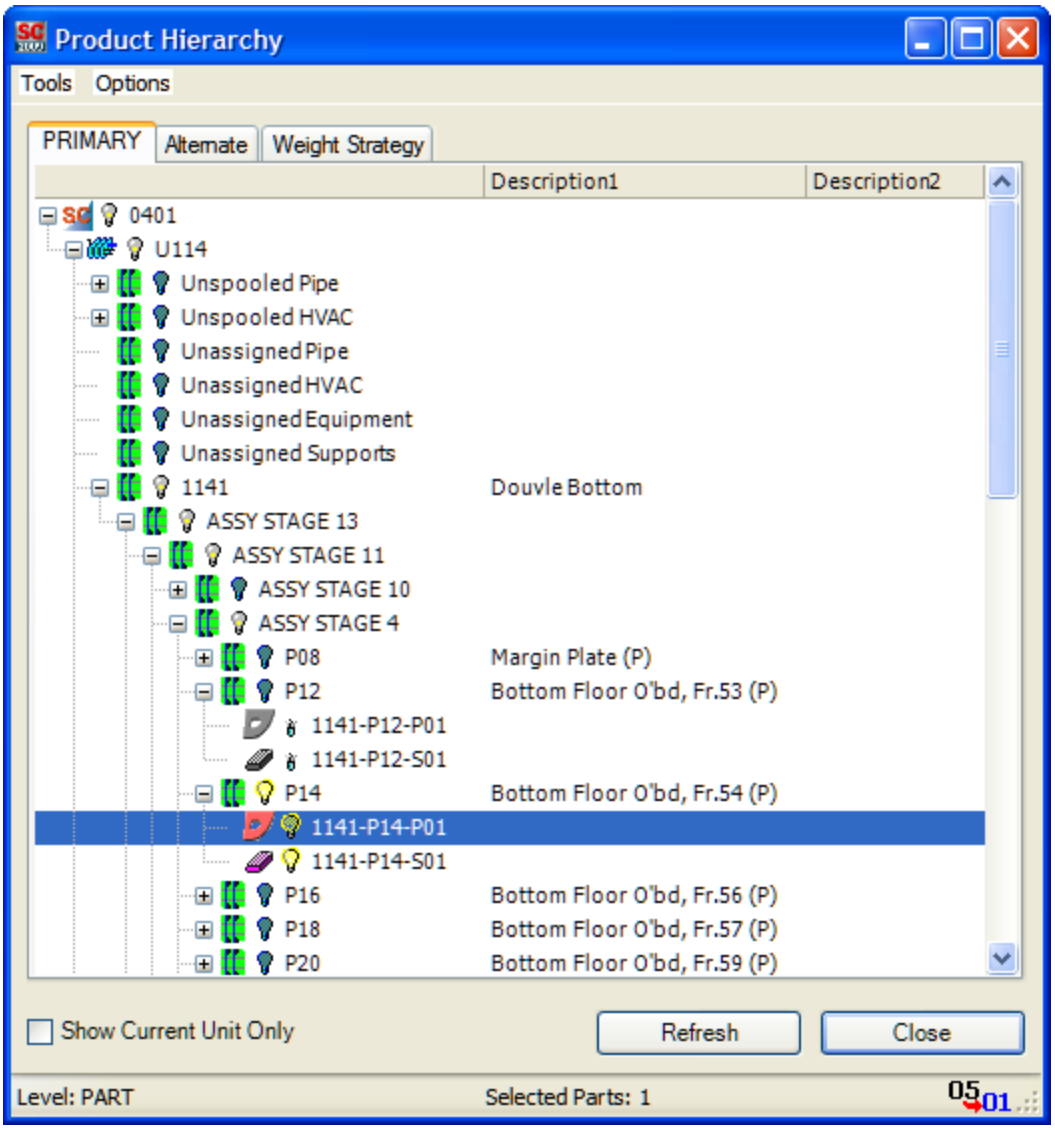
SC Product Hierarchy Menu

The SC Product Hierarchy menu appears in product hierarchy drawings (see Setting up Multiple Product Hierarchies (page 109)).

SC Product Hierarchy > Develop Product Hierarchy

Button .....
Ribbon ..... Structure Production tab > Navigation > Develop Product Hierarchy
Menu ..... SC Producy Hierarchy > Develop Product Hierarchy
Toolbar..... Assembly >
Command ..... SCPRODUCTHIERARCHY
Permissions ... None
Procedure..... Product Hierarchies (page 108)

Opens the Product Hierarchy window, letting you setup assembly levels and assemblies and assign parts to assemblies.



SC Product Hierarchy > Update Drawing

Button .....
Ribbon ..... Structure Production tab > Assembly Drawing panel > Update Drawing
Menu ..... SC Product Hierarchy > Update Drawing
Toolbar..... None
Command ..... SCUPDATEDWG
Permissions ... General > Build Strategy Drawings - Edit
Procedure..... Update Parts in an Assembly Drawing (page 229)

Updates the parts within a product hierarchy drawing.

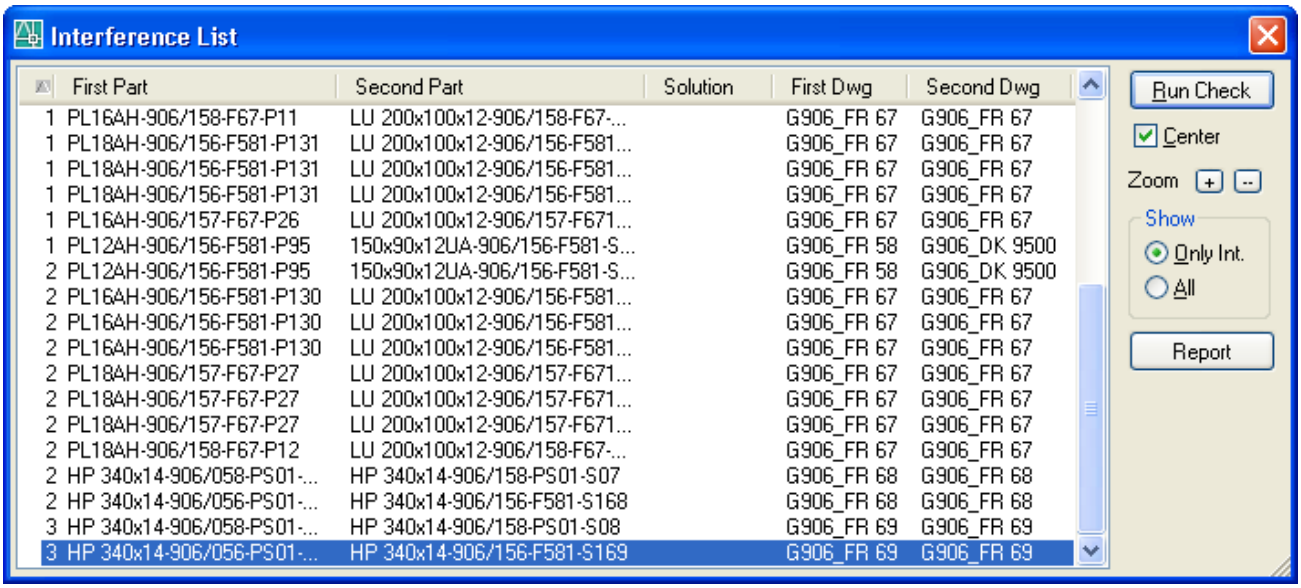
SC Interference Menu

The SC Interference menu appears in interference drawings.

SC Interference > Check Interferences

Button .....
Ribbon ..... Structure Production tab > Modeling/Production Utilities panel > Check Interferences
Menu ..... SC Interference > Check Interferences
Toolbar..... None
Command ..... SCINTERCHECK
Permissions ... General > Edit Interference
Procedure..... Check Interferences (page 200)

Checks the current interference drawing for parts that interfere.



Run Check – Start the interference checking procedure.

SC Interference > Update Drawing

Button .....
Ribbon ..... Structure Production tab > Assembly Drawing panel > Update Drawing
Menu ..... SC Interference > Update Drawing
Toolbar..... None
Command ..... SCUPDATEDWG
Permissions ... General > Interference Drawings - Edit
Procedure..... Update Parts in an Assembly Drawing (page 229)

Updates the parts within a product hierarchy drawing.

SC Plate Nest Menu

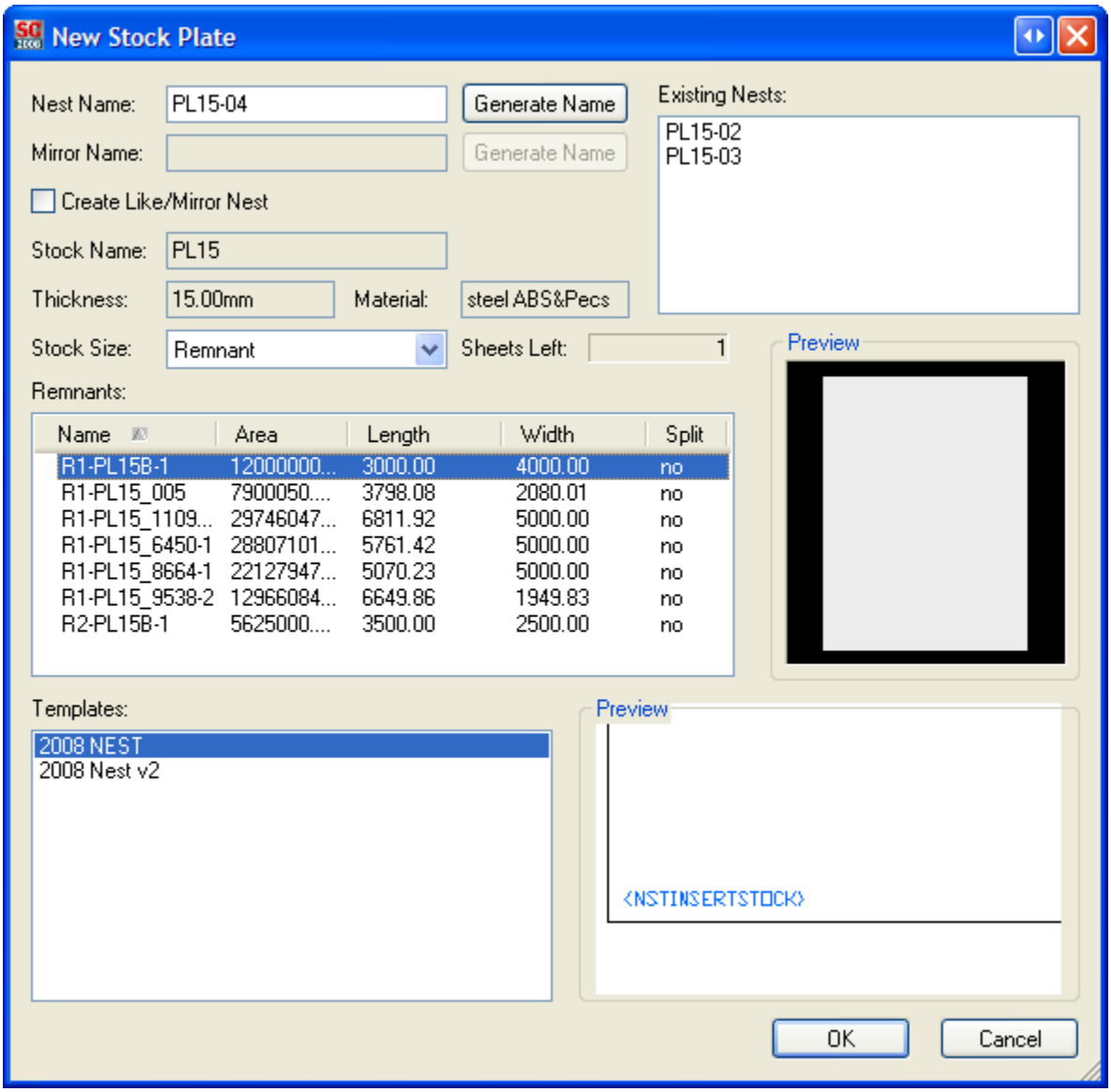
The SC Plate Nest menu appears in nest drawings (see Set Up Nest Drawings and Nests (page 259)).

SC Plate Nest > Nest > New

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > New
Menu ..... SC Plate Nest > Nest > New
Toolbar..... None
Command ..... SCNESTNEW
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Insert a Nest Into a Nest Drawing (page 261)

Inserts a new nest into a nest drawing. Nests are based on a nest template. The template contains keywords. Some of the keywords are replaced with ShipConstructor database data during insertion of a nest; others are replaced at a later stage.

New Stock Plate Window



Nest Name – The function automatically uses the next available number for the nest name.

Create Like/Mirror Nest – Select this to create a like-and-mirror nest.

Stock Name – The name of the stock assigned to the current drawing.

Thickness – The thickness of the stock assigned to the current drawing.

Material – The material of the stock assigned to the current drawing.

Existing Nests – Displays the names of nest plates already in this nest drawing.

Stock Size – Displays the available stock sizes.

Remnants – Lists all available remnants of the same stock. You can use a remnant instead of raw stock for a nest.

Templates – Displays the available templates.

SC Plate Nest > Nest > Delete

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Delete
Menu ..... SC Plate Nest > Nest > Delete
Toolbar..... None
Command ..... SCNESTDELETE
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Delete a Nest (page 281)

Deletes a nest. Pick the nest you want to delete. You can pick on the nest border or the actual plate to select the nest. The color of each of these parts will change to the unassigned nesting colors.

Note: You can also use the AutoCAD **ERASE** command to delete a nest.

SC Plate Nest > Nest > Find

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Find
Menu ..... SC Plate Nest > Nest > Find
Toolbar..... None
Command ..... SCNESTFIND
Permissions ... None
Procedure..... Find a Nest (page 281)

Lets you zoom into any nest in the project.

SC Plate Nest > Nest > Re-Nest

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Re-nest
Menu ..... SC Plate Nest > Nest > Re-nest
Toolbar..... None
Command ..... SCNESTRENEST
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Re-nest a Nest (page 272)

Applies a different nesting algorithm to a set of nested parts and automatically re-nest them (for example, if you are not satisfied with the original nesting).

SC Plate Nest > Nest > Edit Stock Plate

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Edit Stock Plate
Menu ..... SC Plate Nest > Nest > Edit Stock Plate
Toolbar..... None
Command ..... SCNESTEDIT
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Edit the Stock Plate of a Nest (page 281)

Changes the stock plate used for the nest. You can only change the name, stock size, or change between using a full stock plate or a remnant. You cannot change the stock thickness or type of material.

SC Plate Nest > Nest > Plot

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Plot
Menu ..... SC Plate Nest > Nest > Plot
Toolbar..... None
Command ..... SCNESTPLOT
Permissions ... None
Procedure..... Plot Nests (page 294)

Plots nests to a printer.

SC Plate Nest > Nest > Assign to NC-Machine

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Assign to NC-Machine
Menu ..... SC Plate Nest > Nest > Assign to NC-Machine
Toolbar..... None
Command ..... SCNESTASSIGNNCMACHINE
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Assign to NC-Machine (page 284)

Assigns a nest to an NC-Machine. The bridge width for each nest is defined by the NC-Machine assigned to that nest.

SC Plate Nest > Remnant > New

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > New
Menu ..... SC Plate Nest > Remnant > New
Toolbar..... None
Command ..... SCNESTNEWREMNANT
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Create a Remnant (page 285)

Creates a new remnant.

SC Plate Nest > Remnant > Delete

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Delete
Menu ..... SC Plate Nest > Remnant > Delete
Toolbar..... None
Command ..... SCNESTDELETEREMNANT
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Delete a Remnant (page 288)

Deletes a remnant.

Note: You can also use the AutoCAD **ERASE** command to delete a remnant.

SC Plate Nest > Remnant > Edit Remnant

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Edit Remnant
Menu ..... SC Plate Nest > Remnant > Edit Remnant
Toolbar..... None
Command ..... SCNESTEDITREMNANT
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Edit a Remnant (page 287)

Redefines a remnant's shape.

SC Plate Nest > Remnant > Edit Remnant Label

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Edit Remnant Label
Menu ..... SC Plate Nest > Remnant > Edit Remnant Label
Toolbar..... None
Command ..... SCNESTEDITREMNANTLABEL
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Edit a Remnant (page 287)

Changes a remnant's label.

SC Plate Nest > Part > Insert

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Insert
Menu ..... SC Plate Nest > Part > Insert
Toolbar..... Nest >
Command ..... SCNESTINSERTPART
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Insert Parts Into a Nest (page 262)
..... Automatically Insert and Nest Several Parts (page 266)

Inserts parts into a nest, either manually or automatically (if you have a license for AutomaticNest). Parts that have not been inserted into a nest drawing are listed.

SC Plate Nest > Part > Re-insert

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Reinsert
Menu ..... SC Plate Nest > Part > Reinsert
Toolbar..... None
Command ..... SCNESTUPDATEPART
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Re-insert Parts Into a Nest (page 290)

Re-inserts the selected parts using the latest state. A check function is run and will warn the user if any of the parts are out-of-date.

SC Plate Nest > Part > Remove

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Remove
Menu ..... SC Plate Nest > Part > Remove
Toolbar..... None
Command ..... SCNESTREMOVEPART
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Remove Parts from a Nest (page 272)

Un-nests the selected parts. The parts are removed from the nest drawing.

Note: You can also use the AutoCAD **ERASE** command to un-nest a part.

SC Plate Nest > Part > Slide Along

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Slide Along
Menu ..... SC Plate Nest > Part > Slide Along
Toolbar..... Nest >
Command ..... SCNESTSLIDEPART
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Slide a Part Along a Plate Edge or Part Edge (page 275)

The slide function allows you to slide one part along the plate edge or any number of other parts. On the part to slide you have to select two points. The first point is the slide point; the second point indicates a tangent direction. The tangent direction will be aligned with the outside of the other part or the plate edge. A ghost image is displayed around the sliding part to indicate the gap.

SC Plate Nest > Part > Move/Rotate

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Move/Rotate
Menu ..... SC Plate Nest > Part > Move/Rotate
Toolbar..... Nest >
Command ..... SCNESTMOVEPART
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Move or Rotate Parts within a Nest (page 277)

Moves and rotates parts. In some cases it is the easiest to move and rotate the part around until it fits an empty space.

SC Plate Nest > Part > Align

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Align
Menu ..... SC Plate Nest > Part > Align
Toolbar..... None
Command ..... SCNESTALIGNPART
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure.....

Aligns a part with the horizontal or vertical.

SC Plate Nest > Part > Rotate Shortest Dimension

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Rotate Shortest Dimension
Menu ..... SC Plate Nest > Part > Rotate Shortest Dimension
Toolbar..... None
Command ..... SCNESTROTATEPART
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Move or Rotate Parts within a Nest (page 277)

Rotates the part so that it fits into the smallest horizontal or vertical rectangle.

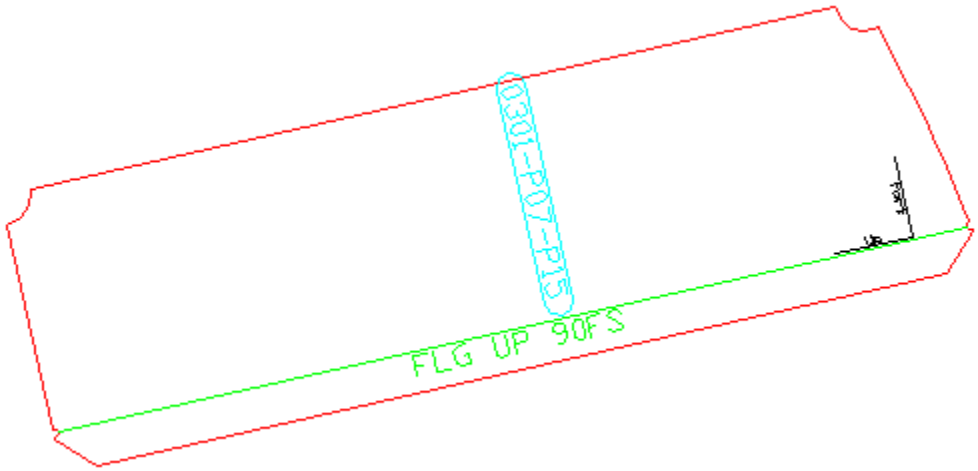


Plate part before Rotate Shortest Dimension

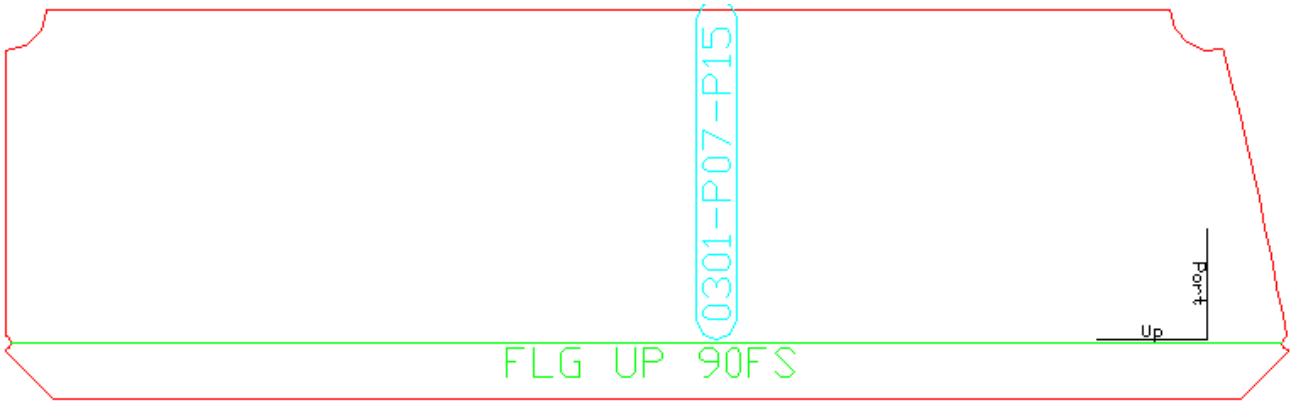
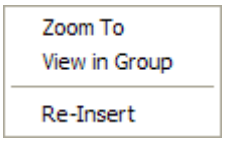


Plate part after Rotate Shortest Dimension

SC Plate Nest > Part > List

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > List
Menu ..... SC Plate Nest > Part > List
Toolbar..... None
Command ..... SCPARTLIST
Permissions ... None
Procedure..... Finding Parts in a Nest Drawing (page 277)

Displays a list of all parts, similar to SC Structure > Parts > List (page 431). However, the right-click menu is slightly different.



SC Plate Nest > Part > Find

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Find
Menu ..... SC Plate Nest > Part > Find
Toolbar..... None
Command ..... SCFINDPARTDWG
Permissions ... None
Procedure..... None

See SC Structure > Parts > Find (page 432).

SC Plate Nest > Part > Snap

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Snap
Menu ..... SC Plate Nest > Part > Snap
Toolbar..... Nest >
Command ..... SCNESTSNAPPART
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Position Parts Next to each other (page 279)

Lets you place a part against the edge of the plate or another part, leaving a specified gap between parts.

SC Plate Nest > Part > Align with H or V

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Align with H or V
Menu ..... SC Plate Nest > Part > Align with H or V
Toolbar..... Nest >
Command ..... SCNESTALIGNPARTHV
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Move or Rotate Parts within a Nest (page 277)

Aligns one side of a part with the horizontal or vertical axis.

SC Plate Nest > Part > List Un-nested Parts

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > List Un-nested Parts
Menu ..... SC Plate Nest > Part > List Un-nested Parts
Toolbar..... None
Command ..... SCNESTLISTUNNESTED
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... List Un-nested Parts in an Assembly (page 278)

List all un-nested parts within an assembly (for example, to ensure that all parts for an assembly are ready in time for production). It will list parts that are not currently assigned to a nest.

SC Plate Nest > Part > Export Nest Parts to DWG

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Export Nest Parts to DWG
Menu ..... SC Plate Nest > Part > Export Nest Parts to DWG
Toolbar..... None
Command ..... SCNESTEXPORTPARTS
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Export Parts to Individual Drawings (page 292)

Creates an individual drawing file for each nest part selected to be exported. Each generated drawing is opened, zoomed to the extents of the objects and unused layers are purged.

SC Plate Nest > Part > Export Nest Parts to DXF

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Export Nest Parts to DXF
Menu ..... SC Plate Nest > Part > Export Nest Parts to DXF
Toolbar..... None
Command ..... SCNESTEXPORTPARTSDXF
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Export Parts to Individual Drawings (page 292)

Creates an individual drawing file for each nest part selected to be exported in DXF format.

SC Plate Nest > Layers > Visibility

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Visibility
Menu ..... SC Plate Nest > Part > Visibility
Toolbar..... Nest >
Command ..... SCNESTLAYER
Permissions ... None
Procedure..... Hide or Show Nested Parts, Revised Parts, or NC Cutting Paths (page 280)

Controls the visibility of the nesting layers.

SC Plate Nest > Layers > Activate All

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Activate All
Menu ..... SC Plate Nest > Layer > Activate All
Toolbar..... None
Command ..... SCNESTLAYERALL
Permissions ... None
Procedure..... Hide or Show Nested Parts, Revised Parts, or NC Cutting Paths (page 280)

Displays all layers in a nest drawing.

SC Plate Nest > Layers > Flip NC-Pyros

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Flip NC-Pyros
Menu ..... SC Plate Nest > Layer > Flip NC-Pyros
Toolbar..... None
Command ..... SCNESTLAYERFLIP
Permissions ... None
Procedure..... Hide or Show Nested Parts, Revised Parts, or NC Cutting Paths (page 280)

Flips between the nesting and NC-Pyros layers. You can flip back and forth. Use this to investigate specific areas of the NC path if problems occur.

SC Plate Nest > Insert Keywords

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Insert Keywords
Menu ..... SC Plate Nest > Insert Keywords
Toolbar..... None
Command ..... SCINSERTKEYWORD
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Insert Keywords into a Nest (page 281)

Lets you insert keywords into a nest. See also SC Plate Nest Template > Insert Keywords (page 526).

SC Plate Nest > Auto Label from BOM Index

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Auto Label from BOM Index
Menu ..... SC Plate Nest > Auto Label from BOM Index
Toolbar..... None
Command ..... SCNESTLABELAUTO
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Labelling Parts (page 278)

Labels the nested parts with their index in the BOM.

SC Plate Nest > Label Properties

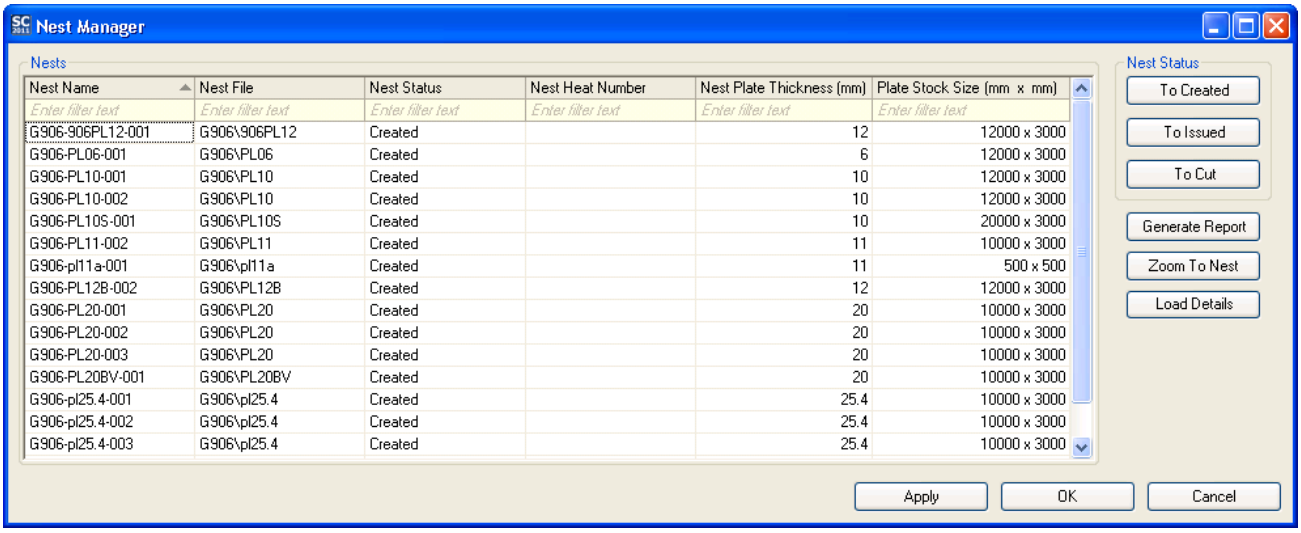
Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Label Properties
Menu ..... SC Plate Nest > Label Properties
Toolbar..... Labels >
Command ..... SCNESTLABELEDIT
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Labelling Parts (page 278)

Edits the properties for a label.

SC Plate Nest > Nest Manager

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Nest Manager
Menu ..... SC Plate Nest > Nest Manager
Toolbar..... None
Command ..... SCNESTMANAGER
Permissions ... None
Procedure..... None

Displays the Nest Manager.


Nest Manager
Nests
Nest Name
Nest File
Nest Status
Nest Heat Number
Nest Plate Thickness (mm)
Plate Stock Size (mm x mm)
G906-906PL12-001 G906\906PL12 Created 12 12000 x 3000
G906-PL06-001 G906\PL06 Created 6 12000 x 3000
G906-PL10-001 G906\PL10 Created 10 12000 x 3000
G906-PL10-002 G906\PL10 Created 10 12000 x 3000
G906-PL10S-001 G906\PL10S Created 10 20000 x 3000
G906-PL11-002 G906\PL11 Created 11 10000 x 3000
G906-pl11a-001 G906\pl11a Created 11 500 x 500
G906-PL12B-002 G906\PL12B Created 12 12000 x 3000
G906-PL20-001 G906\PL20 Created 20 10000 x 3000
G906-PL20-002 G906\PL20 Created 20 10000 x 3000
G906-PL20-003 G906\PL20 Created 20 10000 x 3000
G906-PL20BV-001 G906\PL20BV Created 20 10000 x 3000
G906-pl25.4-001 G906\pl25.4 Created 25.4 10000 x 3000
G906-pl25.4-002 G906\pl25.4 Created 25.4 10000 x 3000
G906-pl25.4-003 G906\pl25.4 Created 25.4 10000 x 3000
Nest Status
To Created
To Issued
To Cut
Generate Report
Zoom To Nest
Load Details
Apply
OK
Cancel

The Nest Manager shows all the nests and lets you enter heat numbers.

Nest Status – Lets you modify nests through the stages from Created to Issued to Cut. Currently the nest status does not have any effect on the parts.

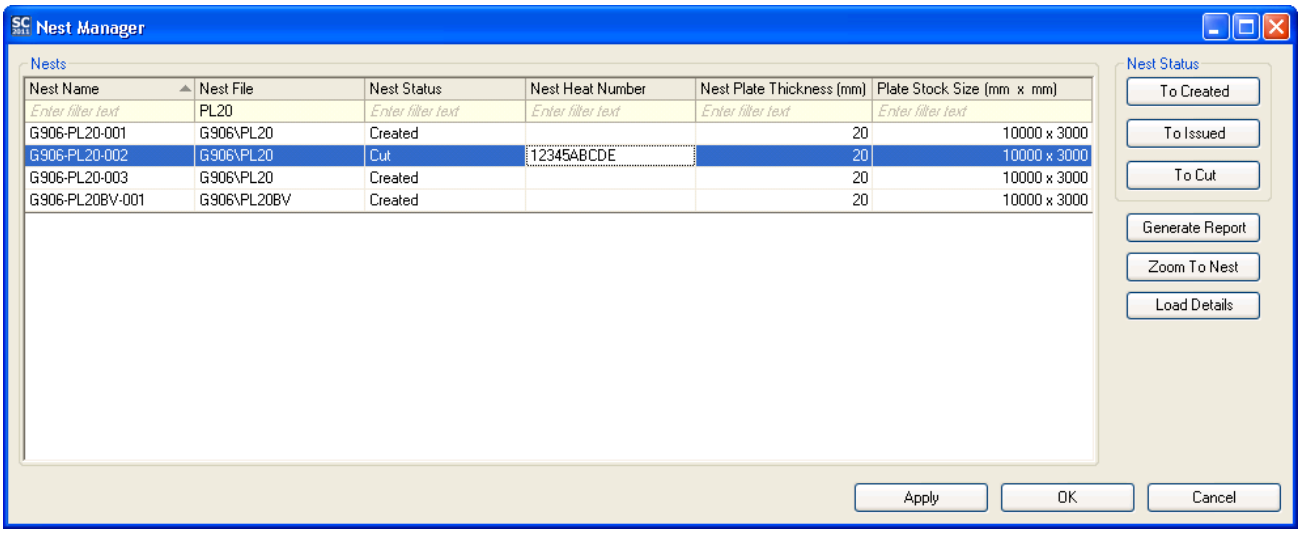
Generate Report – Creates a log file with all the information displayed in the window.

Nest Name	Nest File	Nest Status	Nest Heat Number	Nest Plate Thickness (mm)	Plate Stock Size (mm x mm)
G906-PL20-001	G906\PL20	Created		20	10000 x 3000
G906-PL20-002	G906\PL20	Cut	12345ABCDE	20	10000 x 3000
G906-PL20-003	G906\PL20	Created		20	10000 x 3000
G906-PL20BV-001	G906\PL20BV	Created		20	10000 x 3000

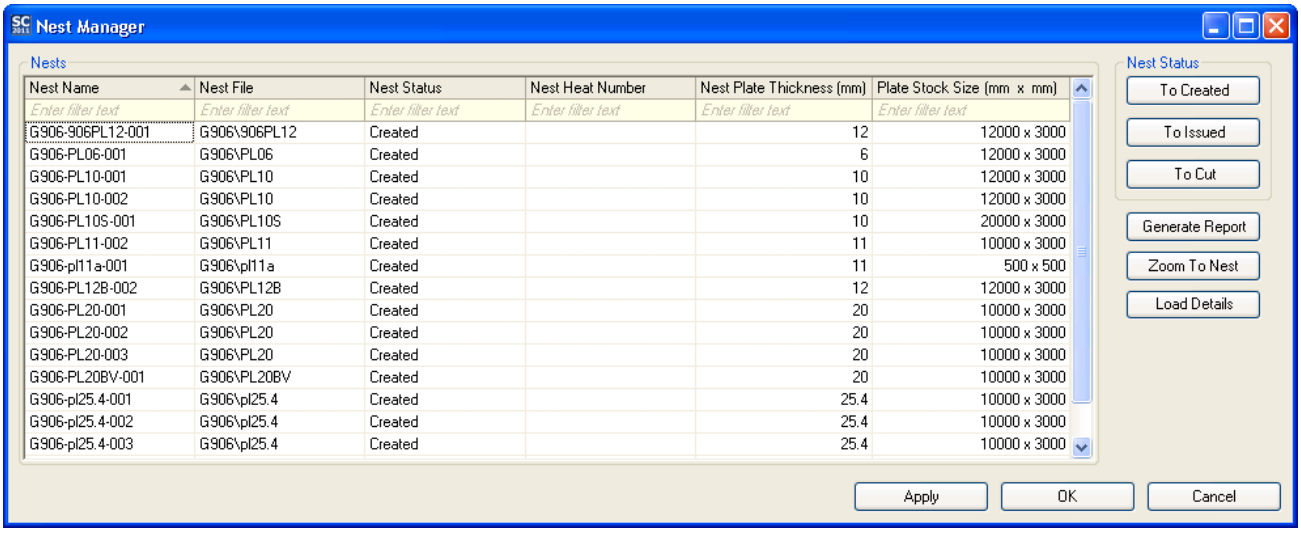
Zoom To Nest – Opens the nest's drawing, and zooms to it.

Load Details – Loading all the nest properties when the window is opened is very time-intensive. To reduce times, only basic properties are loaded initially, and if you want to see more, the Load Details button will then load the unloaded properties.

The columns can be filtered by typing in the textbox just below the column headings.



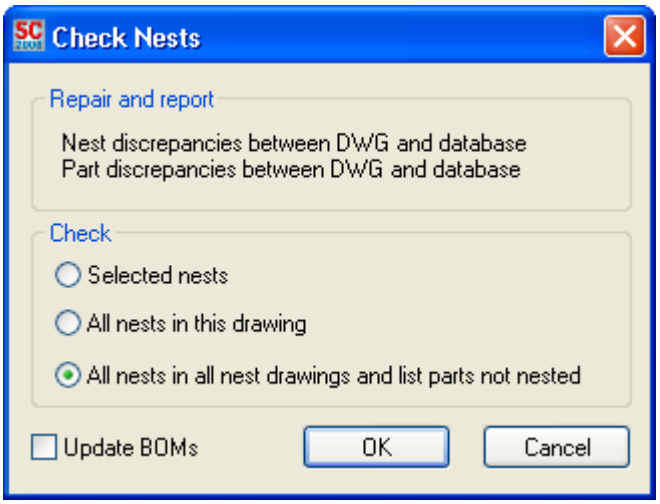
The displayed columns can be modified by right-clicking on a column header and selecting from the list shown.


Nest Manager
Nests
Nest Name
Nest File
Nest Status
Nest Heat Number
Nest Plate Thickness (mm)
Plate Stock Size (mm x mm)
G906-906PL12-001 G906\906PL12 Created 12 12000 x 3000
G906-PL06-001 G906\PL06 Created 6 12000 x 3000
G906-PL10-001 G906\PL10 Created 10 12000 x 3000
G906-PL10-002 G906\PL10 Created 10 12000 x 3000
G906-PL10S-001 G906\PL10S Created 10 20000 x 3000
G906-PL11-002 G906\PL11 Created 11 10000 x 3000
G906-pl11a-001 G906\pl11a Created 11 500 x 500
G906-PL12B-002 G906\PL12B Created 12 12000 x 3000
G906-PL20-001 G906\PL20 Created 20 10000 x 3000
G906-PL20-002 G906\PL20 Created 20 10000 x 3000
G906-PL20-003 G906\PL20 Created 20 10000 x 3000
G906-PL20BV-001 G906\PL20BV Created 20 10000 x 3000
G906-pl25.4-001 G906\pl25.4 Created 25.4 10000 x 3000
G906-pl25.4-002 G906\pl25.4 Created 25.4 10000 x 3000
G906-pl25.4-003 G906\pl25.4 Created 25.4 10000 x 3000
Nest Status
To Created
To Issued
To Cut
Generate Report
Zoom To Nest
Load Details
Apply
OK
Cancel

SC Plate Nest > Check Nests and BOM

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Check Nests and BOM
Menu ..... SC Plate Nest > Check Nests and BOM
Toolbar..... None
Command ..... SCNESTCHECK
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Check Nest Drawings for Errors and Un-nested Parts (page 288)

Checks a nest drawing for errors and updates the BOMs.



SC Plate Nest > Update Nests and BOM

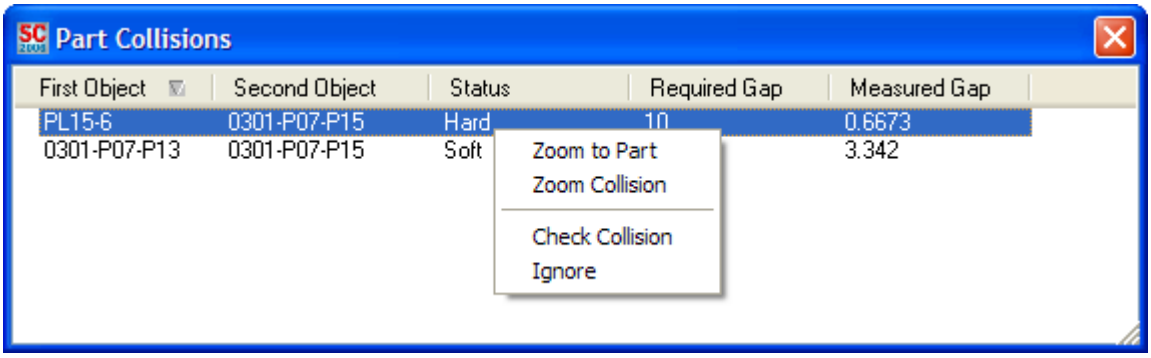
Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel >> Update Nests and BOM
Menu ..... SC Plate Nest > Update Nests and BOM
Toolbar..... None
Command ..... SCNESTUPDATE
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Update Nest Keywords and BOMs (page 289)

Updates the nest keywords and the BOMs for the selected nests (without performing extensive checks like SC Plate Nest > Check Nests and BOM (page 521)).

SC Plate Nest > Nest Collision Check

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Nest Collision Check
Menu ..... SC Plate Nest > Nest Collision Check
Toolbar..... None
Command ..... SCNESTCOLLISIONS
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Check Nests for Collisions (page 289)

Checks if any of the nested parts overlap each other, are too close to each other or are too close to the edge of the plate (based on the part gap defined in Manager for each stock).



Double-clicking on the each collision will highlight the two colliding parts and show circles drawn at the postion of the interference. The stock's Nest Part Offset and Nest Edge Offset are the values that determine if the items collide.

Status

Soft – The two items are within the offset but do not overlap. The Measured Gap indicates how close the items are to each other.

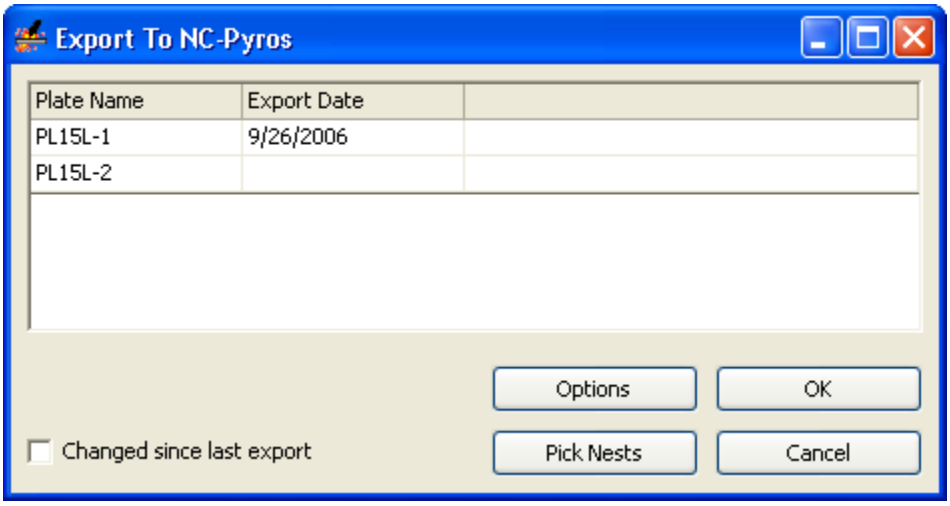
Hard – The items overlap and would cut across each other if the nest was cut.

SC Plate Nest > NC-Pyros > Export To

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Export To
Menu ..... SC Plate Nest > NC-Pyros > Export To
Toolbar..... None
Command ..... SCNESTEXPORTNCPYROS
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Export Nests to NC-Pyros (page 290)

Exports nests to DXF files that you can read into NC-Pyros for creating cutting paths.

Export To NC-Pyros Window

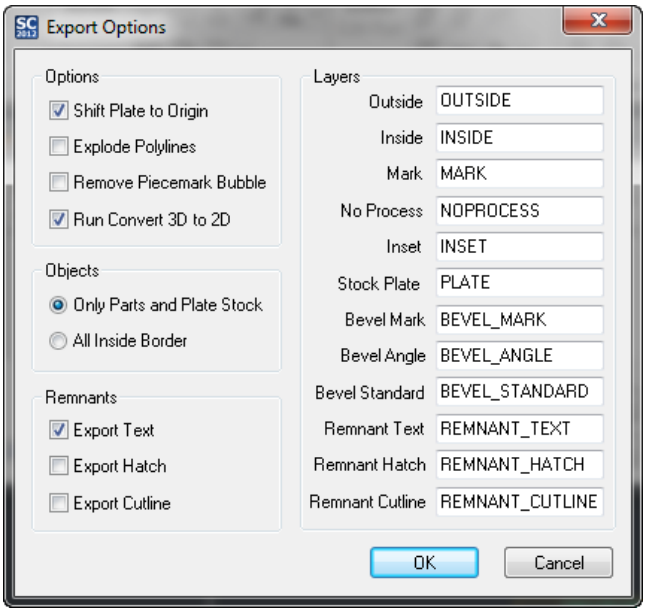


Show only nests from current drawing – If not checked, all nests in the project will be listed. The function will open all drawings and export the nests that have been selected from that drawing.

Changed since last Export – If checked, will only list nests that have been changed or are new since last exporting to NC-Pyros.

Pick Nests – Lets you pick the nests in the drawing to export.

Options – Opens the Export Options window.



Shift Plate to Origin – Check this option if you want to move the lower left edge of the nest plate to 0,0 in the exported drawing. While NC-Pyros can do this for you automatically, other nesting software may not. If you intend to use the Import From command, then do not use this option because the Import From will show all nests at the origin.

Explode Polylines – For NC software that cannot read polylines.

Remove Piecemark Bubble – This lets you remove the bubble when the parts contain them in the model but you do not want them marked.

Run Convert 3D to 2D – NC code generation software may not handle objects with an extrusion direction. This command ensures that the extrusion direction is removed. Evidence of this case is when geometry is placed in an unexpected location in the NC code generation program.

Only Parts and plate stock – Exports only the objects inside the plate stock area. Use this option when you have some objects in the header of the nest that are in a processed color.

All inside border – Export every object that is inside the nest border.

Remnants – The various components of a remnant can be exported individually and placed on unique layers.

Export Text – Export the remnant's name text

Export Hatch – Export the remnant's hatch

Export Cutline – Export the remnant's cutline

Layers – NC-Pyros can differentiate processes by color or by layer. Colors or layers can be used to distinguish between outside, inside, marking, no process, inset, plate, bevel marking, bevel angles and bevel standard angles. The colors for the various objects types are settable in the Manager > General > Project Settings > Structure Colors > NC-Pyros Colors. The layer names of the various objects are separated using the layer name specified here.

Outside – Outside cuts

Inside – Holes in parts

Mark – Marklines and text on parts

No Process – Objects not to be processed but are exported

Inset – Parts inside holes of parts

Plate – The stock plate border

Bevel Mark – Marking for bevel cutting. Bevel Standards are labeled marked on this layer.

Bevel Angle – Variable and constant bevel angles are placed as text objects on this layer with XDATA that indicates detailed information about the cut.

Bevel Standard – Bevel cuts made from Bevel Standards.

Remnant Text – Text with remnant name.

Remnant Hatch – Hatch representing remnant area.

Remnant Cutline – Line representing where to cut to create the remnant.

SC Plate Nest > NC-Pyros > Import From

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Import From
Menu ..... SC Plate Nest > NC-Pyros > Import From
Toolbar..... None
Command ..... SCNESTIMPORTNCPYROS
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Import Cutting Paths into Nests (page 292)

Imports a DXO file (containing NC cutting paths and direction arrows) generated from NC-Pyros into a nest drawing. The generated path is inserted on top of the nest and is place in an AutoCAD block. You may want to show the path the NC machine follows when cutting a nest on the plot of the nest for the machine operator.

SC Plate Nest > Bridge > New

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > New
Menu ..... SC Plate Nest > Bridge > New
Toolbar..... Nest >
Command ..... SCNESTNEWBRIDGE
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Insert a Bridge (page 284)

Lets you insert a bridge between two parts. A bridge connects two parts so that they do not drop or tilt in the NC-Machine. Bridges also reduce the number of pierces of the cutting head thereby reducing wear on it. The width of the bridge is set for each plate stock in Manager. See Structure Stock Catalog (page 343).

SC Plate Nest > Bridge > Delete

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Delete
Menu ..... SC Plate Nest > Bridge > Delete
Toolbar..... None
Command ..... SCNESTDELETEBRIDGE
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... Delete Bridges (page 285)

Deletes selected bridges in a nest drawing.

Note: You can also use the AutoCAD **ERASE** command to delete a bridge.

SC Plate Nest > Bridge > Update

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest panel > Update
Menu ..... SC Plate Nest > Bridge > Update
Toolbar..... None
Command ..... SCNESTUPDATEBRIDGE
Permissions ... Structure > Plate Nest > Plate Nests - Edit
Procedure..... None

Updates a bridge, if the bridge width has changed in Manager.

SC Plate Nest Template Menu

The SC Plate Nest Template menu is available within nest template drawings. See Set Up Nest Templates (page 255).

SC Plate Nest Template > Define Border

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest Template panel > Define Border
Menu ..... SC Plate Nest Template > Define Border
Toolbar..... None
Command ..... SCNESTNEWTEMPLATE
Permissions ... None
Procedure..... Define a Nest Template Border (page 257)

Uses a closed polyline to define a border for the nest template.

SC Plate Nest Template > Insert Keywords

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest Template panel > Insert Keywords
Menu ..... SC Plate Nest Template > Insert Keywords
Toolbar..... None
Command ..... SCINSERTKEYWORD
Permissions ... None
Procedure..... Insert Keywords into a Nest Template (page 258)

Lets you insert keywords into a nest template.

Keyword	Description
<COMPANY NAME>	Company name from Project Settings
<FILE NAME>	The file name only
<FILEPATH>	The complete path and file name
<HULL NUMBER>	Hull number from Project Settings
<JOB NUMBER>	Job number from Project Settings
<LAST UPDATE DATE AND TIME>	The date and time of the last update of this drawing
<LAST UPDATE DATE>	The date of the last update of this drawing
<NEST AREA>	Area of the nest
<NEST CUT DISTANCE>	Combined distance of all cuts made
<NEST CUT FEED>	Cutting speed of the NC Machine used
<NEST DATE>	Date of creation
<NEST ESTIMATED TIME>	Estimated time to cut and mark this nest, based on cut/mark/fast feeds and distances, and piercing time
<NEST EXPORT DATE>	Date when the nest was exported
<NEST EXPORT TIME>	Time when the nest was exported
<NEST EXPORT USER>	The user who exported the nest
<NEST FAR FINISHES>	Common finishes for far side markings
<NEST FAST FEED>	Fast travel speed of the NC Machine used
<NEST FAST TRAVEL DISTANCE>	Combined distance of all fast travel made
<NEST FINISHES>	Common finishes for both far and mark side
<NEST HEAT NUMBER>	Heat number of the plate stock used
<NEST ISSUE DATE>	Date when the nest was issued
<NEST ISSUE TIME>	Time when the nest was issued
<NEST ISSUE USER>	The user who issued the nest
<NEST LENGTH>	Length of the nest
<NEST LIKE/MIRROR>	The LIKE alias if a single nest, or the LIKE/MIRROR alias if a Like/Mirror nest
<NEST MARK DISTANCE>	Combined distance of all markings made
<NEST MARK FEED>	Marking speed of the NC Machine used
<NEST MARK FINISHES>	Common finishes for mark side markings
<NEST NAME>	Name of the nest
<NEST NCCODE GENERATE DATE>	Date when the nest's NCCode was generated
<NEST NCCODE GENERATE TIME>	Time when the nest's NCCode was generated
<NEST NCCODE GENERATE USER>	The user who created the nest's NCCode
<NEST NCCUT DATE>	Date when the nest was cut
<NEST NCCUT TIME>	Time when the nest was cut
<NEST NCCUT USER>	The user who cut the nest
<NEST NC-MACHINE NAME>	Name of the NC Machine used
<NEST NUMBER OF PIERCES>	Amount of pierces on the nest
<NEST PART AREA>	Area of all parts assigned to the nest
<NEST PART WEIGHT>	Weight of all parts assigned to the nest
<NEST REMNANT LIST>	Lists of all remnants created from this of the nest

<NEST REMNANT NAME>	Name of the remnant this nest was created from
<NEST REMNANT WEIGHT>	Weight of all remnants created from this nest
<NEST SCRAP UTILIZATION>	Percentage of the area that is not parts over the area of the item used to create the nest
<NEST SCRAP WEIGHT>	Remaining weight after subtracting part and remnant weight from the nest stock weight
<NEST SIZE>	Length x Width of the nest
<NEST STOCK AREA>	Area of the item used to create the nest
<NEST STOCK LENGTH>	Length of the item used to create the nest
<NEST STOCK NAME>	Name of the item used to create the nest (remnant name if created from a remnant, or the plate stock name if created from raw stock)
<NEST STOCK PLATE LOCATION>	Indicator keyword for the location of the stock plate in the nest template drawing
<NEST STOCK SIZE>	Length x Width of the item used to create the nest
<NEST STOCK UTILIZATION>	Percentage of the area that is parts and remnants over the area of the item used to create the nest
<NEST STOCK WEIGHT>	Weight of the item used to create the nest
<NEST STOCK WIDTH>	Width of the item used to create the nest
<NEST TIME>	Time of creation
<NEST USER>	The user who created the nest
<NEST UTILIZATION>	Percentage of the area of all parts over the area of the item used to create the nest
<NEST WEIGHT>	Weight of the nest
<NEST WIDTH>	Width of the nest
<PLATE STOCK AREA>	Area of the original plate stock
<PLATE STOCK LENGTH>	Length of the original plate stock
<PLATE STOCK MATERIAL>	Material of the original plate stock
<PLATE STOCK NAME>	Name of the original plate stock
<PLATE STOCK SIZE>	Length x Width of original plate stock
<PLATE STOCK THICKNESS>	Thickness of the plate stock used
<PLATE STOCK WEIGHT>	Weight of the original plate stock
<PLATE STOCK WIDTH>	Width of the original plate stock
<PROJECT DESCRIPTION>	Description of the project from Project Settings
<PROJECT NAME>	Name of the project from Project Settings
<USER>	Current user

SC Nest Template > Insert BOM Table

Button .....
Ribbon ..... Structure Production tab > BOM panel > Insert BOM Table
Menu ..... SC Plate Nest Template > Insert Keywords
Toolbar..... None
Command ..... SCINSERTBOM
Permissions ... None
Procedure..... None

Lets you insert a Part Bill of Materials into a nest template. You must create a BOM definition in Manager before inserting a table (See General > Production Output > Bill Of Materials (page 339)).

SC Plate Nest Template > Update from a Previous Version

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest Template panel > Update from a Previous Version
Menu ..... SC Plate Nest Template > Update from a Previous Version
Toolbar..... None
Command ..... SCNESTUPDATETEMPDWG
Permissions ... None
Procedure.....

Updates the template drawing from a Previous Version (ShipConstructor2005 and earlier) to the format used by ShipConstructor.

SC Plate Nest Template > Check Drawing

Button .....
Ribbon ..... Nest and Profile Plots tab > Plate Nest Template panel > Check Drawing
Menu ..... SC Plate Nest Template > Check Drawing
Toolbar..... None
Command ..... SCNESTCHECKTEMPDWG
Permissions ... None
Procedure..... Check Drawing Validity (page 259)

Runs a check on the template drawing to confirm that it has the correct number of necessary objects inserted, and can be used for nesting.

SC Profile Plots Menu

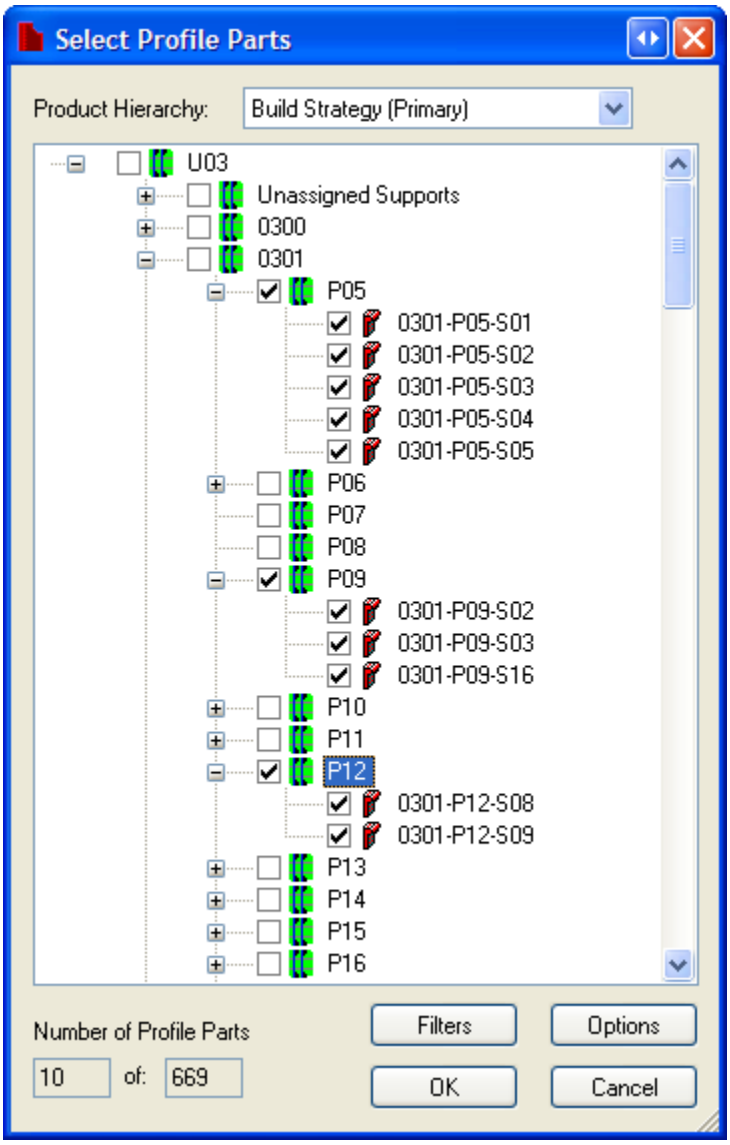
The SC Profile Plots menu is available within profile plot drawings.

SC Profile Plots > Insert Profile Plots

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Plots panel > Insert Profile Plots
Menu ..... SC Profile Plots > Insert Profile Plots
Toolbar..... None
Command ..... SCINSERTSTIFFENERS
Permissions ... None
Procedure..... Insert Profile Plots Into a Profile Plot Drawing (page 301)

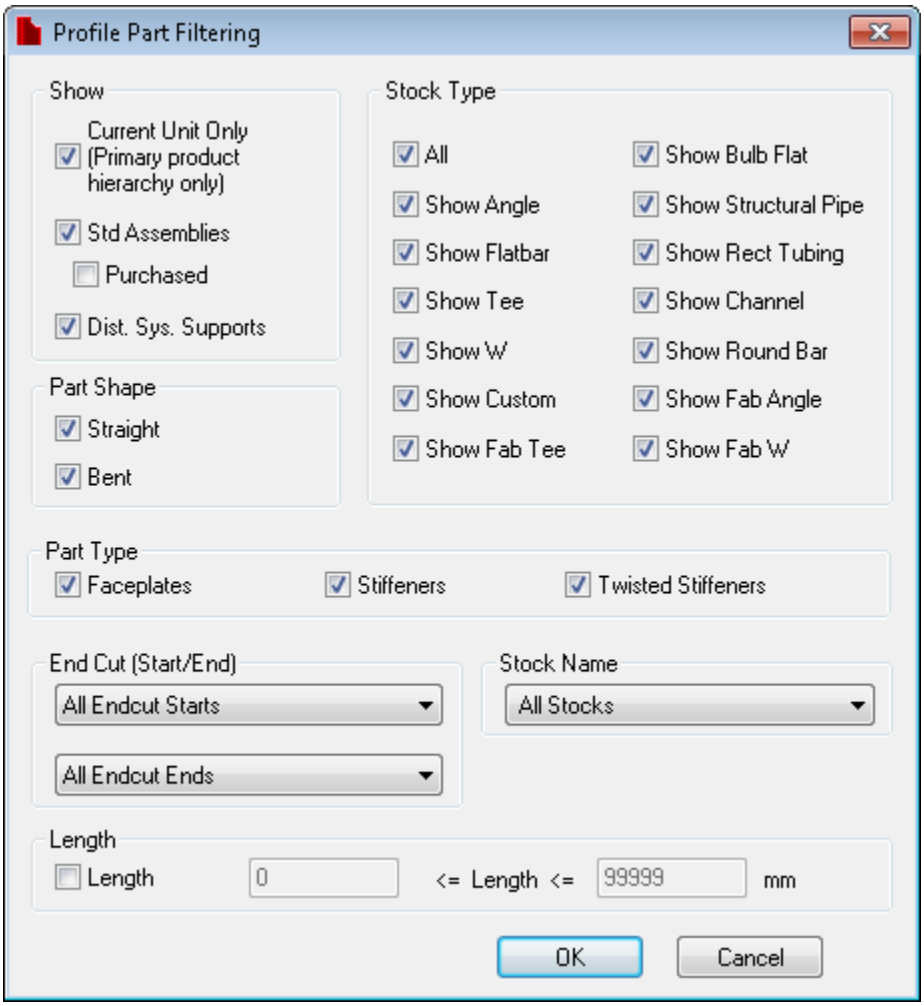
Inserts profile plots into the profile plot drawing.

Select Stiffener Window



Product Hierarchy – The product hierarchy to display in the tree. For the non-primary hierarchies the whole project will be shown. For the primary hierarachy, the Current Unit Only filter is used.

Stiffener Filtering Window



Allows the user to filter the stiffeners displayed in the Stiffener Select Window by the following parameters:

Current Unit Only (Primary product hierarchy only) – This option minimizes the amount of information in the tree, speeding up the window, and is especially useful for large projects.

Std Assemblies – If checked, the filter will display assemblies from standard assemblies, and their parts.

Purchased – If checked, Standard Assemblies tagged as “Purchased” will also be displayed.

Dist. Sys. Supports – If checked, Distributed Systems Supports are displayed.

Part Shape – Bent or Straight

Stock Type – Major stock type.

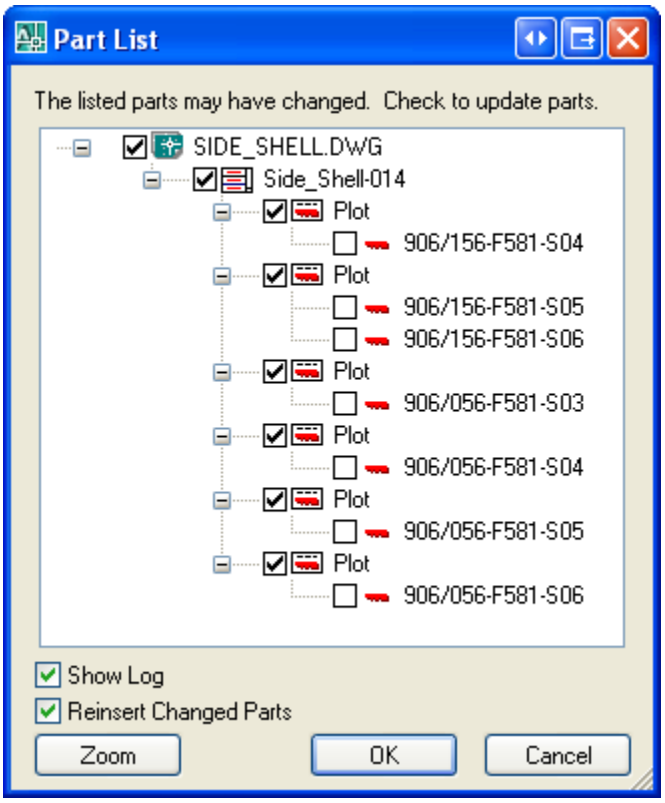
Part Type – Faceplates, Stiffeners and Twisted Stiffeners.

End Cuts – Start and End endcut.

Stock Name – A specific stock.

Length – A specific range of lengths.

Profile Plot Update Window



Allows the user to select which profile plots they would like to update if parts within those plots have been changed in any way.

Show Log – Selects whether a log file will be displayed containing a list of all the major changes made during the update of the selected profile plots.

Reinsert Changed Parts – Selects whether or not the selected parts will be re-inserted.

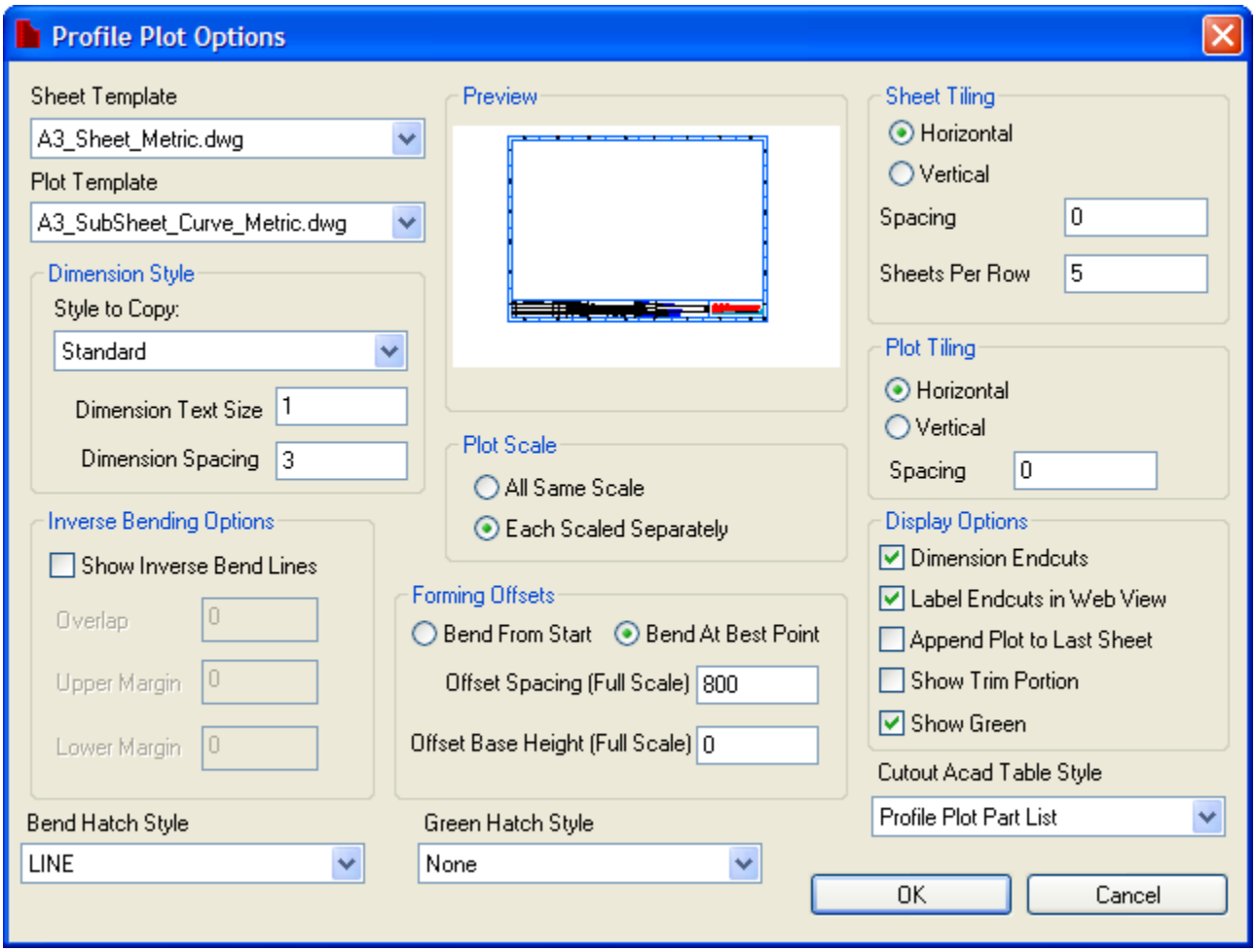
Zoom – Zooms the current view to the selected drawing, sheet, plot or part.

SC Profile Plots > Options

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Plots panel > Options
Menu ..... SC Profile Plots > Options
Toolbar..... None
Command ..... SCSTIFFPLOTOPT
Permissions ... None
Procedure..... Set Up Profile Plots (page 300)

Opens the Stiffener Plot Options window, letting you set up profile plots.

Profile Plot Options Window



Sheet Template – The large template drawing is set up to hold the individual Profile Plots. Customize this template to fit your needs. See Profile Sheet Template (page 296).

Plot Template – A template designed to display one profile and all the related data. Customize this template to fit your needs. See Profile Plot Template (page 297).

Dimension Style – Sets up specific parameters for the dimensions.

Style to Copy – Every time you add plots to a drawing, ShipConstructor has to create a dimension style depending on how the profiles will fit inside the templates (for example, a scaling factor). In general you will first adjust a dimension style of your choice and then tell ShipConstructor to use the settings of that style as a basis for the new style.

Dimension Text Size – Sets the text size independent of the dimension style selected. These units are based on the size of the templates you use. If you set up your templates to be the size of paper, the dimension size will be in paper size units.

Dimension Spacing – Sets the spacing of the dimension from the object. These units are based on the size of the templates you use. If you set up your templates to be the size of paper, the dimension spacing will be in paper size units.

Inverse Bending Options – Settings that control the inverse bending layout.

Show Inverse Bend Lines – Selects whether or not inverse bend lines will be displayed.

Overlap – Distance in ship (model space) coordinates to overlap the inverse bend lines if multiple lines are needed.

Upper Margin – The smallest distance that an inverse bend line will come to the upper edge of the stiffener.

Lower Margin – The smallest distance that an inverse bend line will come to the lower edge of the stiffener (mold line).

> The sum of the upper and lower margins should not exceed the height of the inverse bend view. Otherwise, the height will represent the web height if the stiffener is bending in the plane of the web or the flange width.

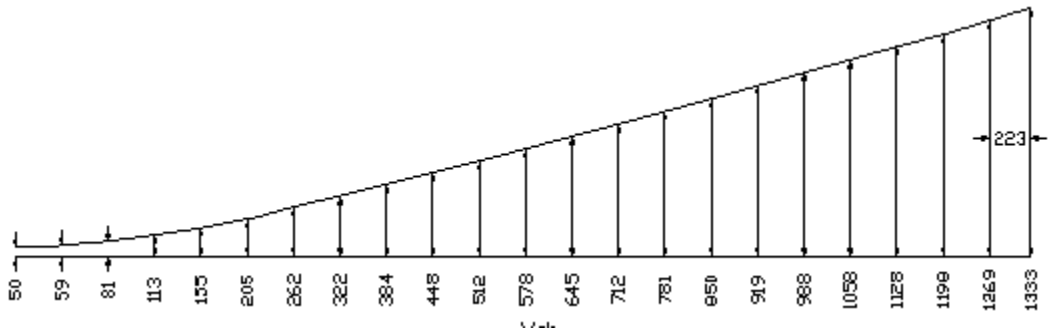
Bend Hatch Style – The hatch style that will be used when hatching the bent area of the profile part. Select None if no hatching is desired.

Green Hatch Style – The hatch style that will be used when hatching the green area of the profile part. Select None if no hatching is desired.

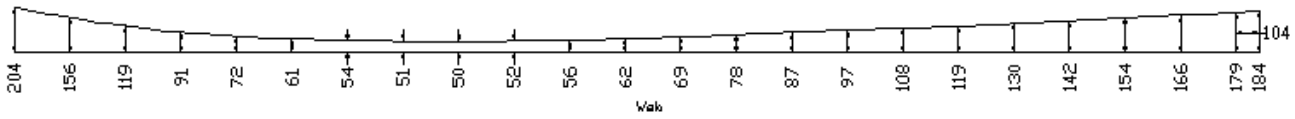
Plot Scale – Selects if you would like to use the scale of the longest stiffener for all profile plots created during the current insertion operation or use a separate scale for each plot.

Forming Offsets – Settings that control the options for the offsets for curved profiles.

Bend from Start – Sets the start of the profile horizontal and lets the profile curve up or down from there. This can result in a very tall plot.



Bend at Best Point – This will rotate the profile so that it will fit into a rectangle of minimum height.



Offset Spacing (Full Scale) – Sets the spacing of the forming offset lines in the original scale of the stiffener.

Offset Base Height (Full Scale) – Sets the minimum forming offset dimension line height in the original scale of the stiffener.

Sheet Tiling – Specifies the direction and spacing of the tiling when inserting multiple profile plot sheets into the drawing during one insertion operation.

Sheets Per Row – The maximum number of sheets that will be placed in a row before a new row is created when inserting multiple profile plot sheets into the drawing during one insertion operation.

Plot Tiling - Specifies the direction and spacing of the tiling when inserting multiple profile plots into a profile plot sheet during one insertion operation.

Dimension Endcuts - Selects whether or not the endcuts are dimensioned in the profile plot.

Label Endcuts in Web View - Selects whether or not the endcuts are labeled in the profile plot.

Append Plot to Last Sheet – When selected subsequent profile plots will be inserted into any available space in the profile plot sheets that already reside in the drawing. If there is no room in any of the existing sheets then a new sheet will be inserted into the drawing and the remaining profile plots will be added to it. When not selected new sheets will be added to the drawing regardless to if there is still room in the existing sheets in the drawing.

Show Trim Portion – Selects whether or not to show the outline of the square end of the stiffener.

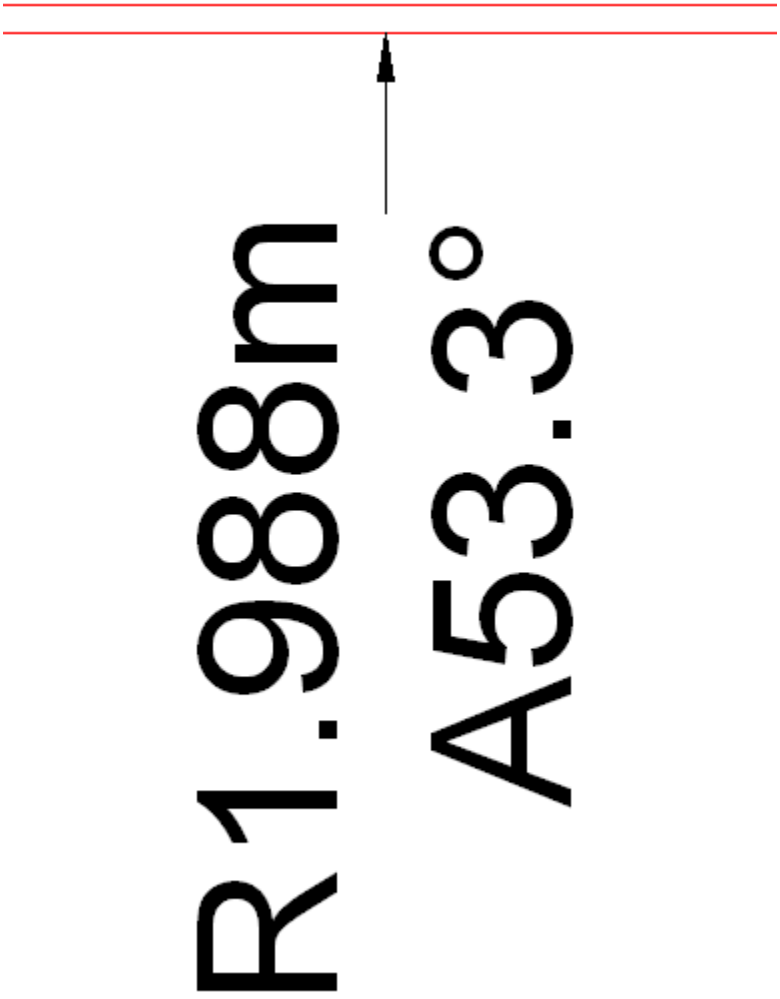
Show Green – Selects whether or not to show the outline of the green part of the stiffener.

Cutout Acad Table Style – Selects the Acad table style that will be used for the cutouts table in this profile plot.

Profile Plot Output

Bending information – The bending information shown below.

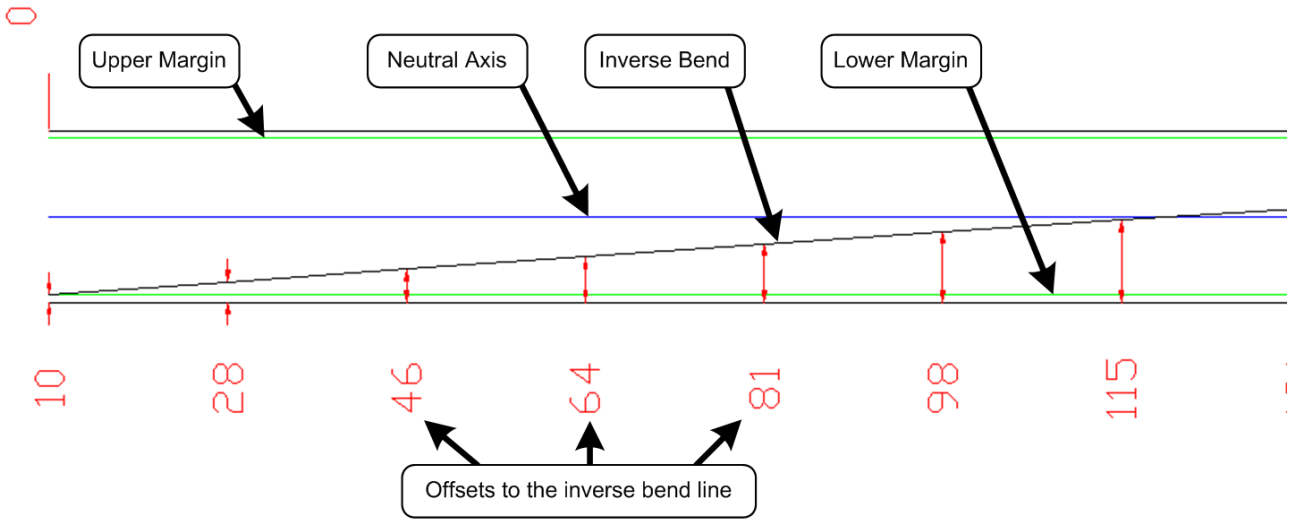
The dimension is positioned at the start tangent (left side). The end tangent is not indicated. For non-radial bends, ShipConstructor uses knuckling (bends of R0). The Manager Preference setting Faceplate Bend Tolerance is used to determine how much to knuckle. Since a dimension style is set up for the scale of every plot, you can manually dimension the offset curve if you prefer.



R1.988 indicates the inside radius of bend. This is in the Report Units set up in Manager.

A53.3º indicates the enclosing angle of the bend

Inverse Bend Information
Inverse bending is used to bend stiffeners to their final shape by drawing curved inverse bending lines onto the stiffeners and bending the stiffeners until these lines are straight. Inverse bending information only appears for curved or twisted stiffeners.



The distance between offset dimensions for the inverse bend is taken from the Offset Spacing field in the Forming Offsets.

The Upper and Lower margin lines are placed on the Margin layer.

The Neutral Axis is placed on the Neutral Axis layer.

The Inverse bend line is placed on the Inverse Bend Lines layer.

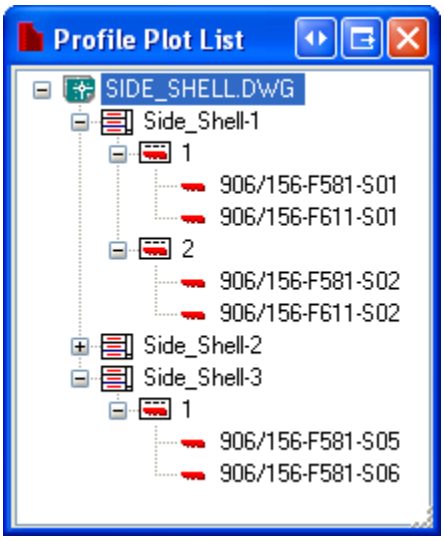
The offsets are placed on the _PRD_DIM layer.

SC Profile Plots > Zoom

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Plots panel > Zoom
Menu ..... SC Profile Plots > Zoom
Toolbar..... None
Command ..... SCZOOMTOPLOT
Permissions ... None
Procedure..... Zoom Into a Profile Plot Within a Profile Plot Drawing (page 304)

Lets you easily zoom in to a specific profile plot or profile plot sheet within a profile plot drawing.

Profile Plot List Window

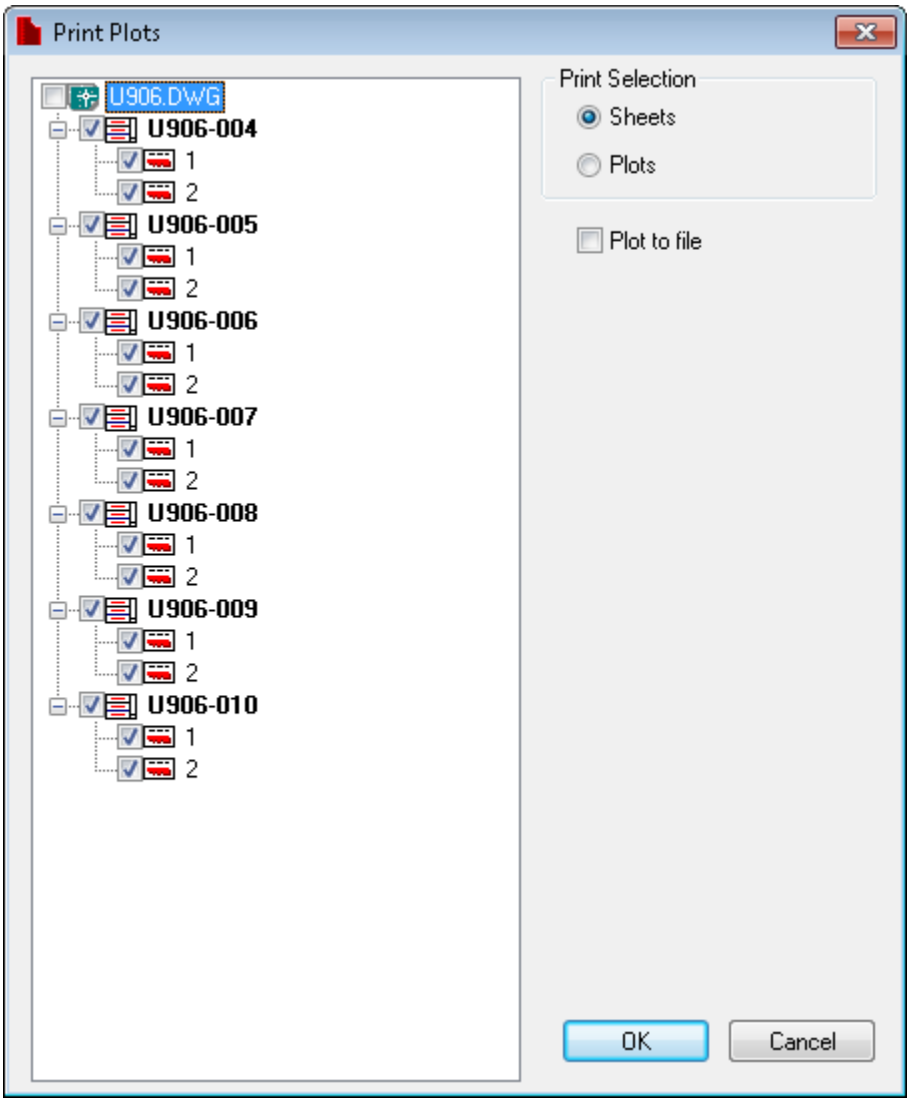


SC Profile Plots > Print Plots

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Plots panel > Print Plots
Menu ..... SC Profile Plots > Print Plots
Toolbar..... None
Command ..... SCSUBPLOTPRINT
Permissions ... None
Procedure..... Print Profile Plots (page 304)

Lets you print all or some profile plots to a printer.

Print Plots Window



Sheets – Check this option if you want to print the profile plot sheets as well as the profile plots.

Plots – Check this option if you want to print the profile plots without printing the profile plot sheets.

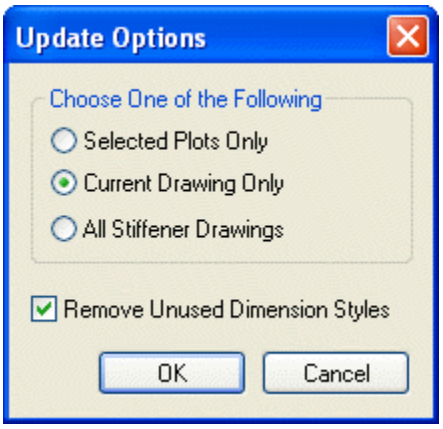
Plot to file – Select this if you want to generate a set of files instead of printing to a device. The filenames will use the ones listed in the Print Plots list.

SC Profile Plots > Check Profile Drawings

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Plots panel > Check Profile Drawings
Menu ..... SC Profile Plots > Check Profile Drawings
Toolbar..... None
Command ..... SCCHECKPLOTS
Permissions ... None
Procedure..... Check Profile Plots (page 304)

Checks profile plots in the project database against profile plot drawings and corrects any errors (for example, any missing profile plot drawings that someone may have deleted manually without using ShipConstructor).

Update Options Window



Selected Plots Only – Lets you select the plots within this drawing.

Current Drawing Only – Checks all plots within this drawing.

All Stiffener Drawings – Checks all plots in all profile plot drawings.

Remove Unused Dimension Styles – Removes dimension styles not used anymore (you may have deleted specific plots).

SC Profile Plots > Update All BOMs

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Plots panel > Update All BOMs
Menu ..... SC Profile Plots > Update All BOMs
Toolbar..... None
Command ..... SCUPDATEBOMS
Permissions ... None
Procedure..... Update Profile Plot BOMs (page 305)

Updates all the BOMs within the current profile plot drawing so they reflect the current information from the plotted parts.

SC Profile Plot Template Menu

The SC Profile Plot Template menu appears within profile plot template drawings (see Set Up Profile Plot Templates (page 296)).

SC Profile Plot Template > Draw Border

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Plot Template panel > Draw Border
Menu ..... SC Profile Plot Template > Draw Border
Toolbar..... None
Command ..... SCPROFILEPLOTSETPLOT
Permissions ... None
Procedure..... Define the Drawing Border of a Plot Template (page 298)

Defines the area of the drawing.

SC Profile Plot Template > Draw Web View Area

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Plot Template panel > Draw Web View Area
Menu ..... SC Profile Plot Template > Draw Web View Area
Toolbar..... None
Command ..... SCPROFILEPLOTSETWEB
Permissions ... None
Procedure..... Define the Web View Area of a Plot Template (page 298)

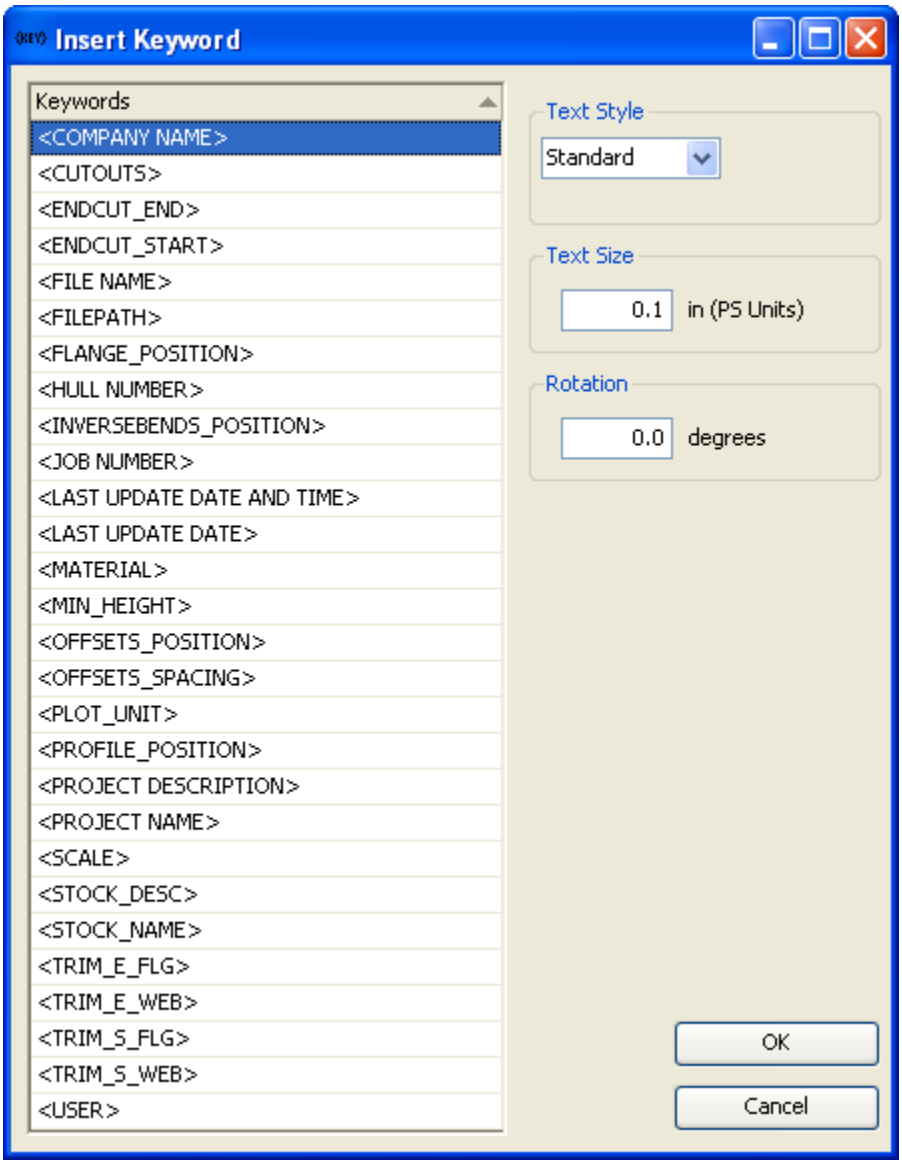
Defines the area (within each profile plot drawing) that will contain profile plots.

SC Profile Plot Template > Insert Keywords

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Plot Template panel > Insert Keywords
Menu ..... SC Profile Plot Template > Insert Keywords
Toolbar..... None
Command ..... SCINSERTKEYWORD
Permissions ... None
Procedure..... Insert Keywords into a Profile Plot Template (page 298)

Lets you insert a keyword into a profile plot template.

Insert Keyword Window



Keywords

Keyword	Description
<COMPANY_NAME>	The value from Project Settings
<CUTOUTS>	The upper left corner of the table of cutouts
<ENDCUT_END>	The endcut at the end of the stiffener
<ENDCUT_START>	The endcut at the start of the stiffener
<FILE NAME>	The file name only
<FILEPATH>	The filepath
<FLANGE_POSITION>	The lower left
<HULL NUMBER>	The name of the hull
<INVERSEBENDS_POSITION>	The lower left corner of the inverse bend view
<JOB NUMBER>	The value from Project Settings
<LAST UPDATE DATE AND TIME>	The date and time when the plot was last updated
<LAST UPDATE DATE>	The date when the plot was last updated
<MATERIAL>	The material and grade of stock for the parts in the plot
<MIN_HEIGHT>	The minimum height for the offsets dimensions used in the offsets view
<OFFSETS_POSITION>	The lower left corner of the offsets view
<OFFSETS_SPACING>	The offsets spacing value used to dimension the offsets view
<PLOT_UNIT>	The common unit of the parts in the plot. If the parts belong to different units then 'Varies' will be displayed
<PROFILE_POSITION>	The lower left corner of the profile cross-section view
<PROJECT DESCRIPTION>	The value from Project Settings
<PROJECT NAME>	The value from Project Settings
<SCALE>	The scale of the plot relative to the full size stiffener
<STOCK_DESC>	The Description field of the stock for the stiffeners in the plot
<STOCK_NAME>	The stock name of the stiffeners in the plot
<TRIM_E_FLG>	The trim on the stiffener's end flange
<TRIM_E_WEB>	The trim on the stiffener's end web
<TRIM_S_FLG>	The trim on the stiffener's start flange
<TRIM_S_WEB>	The trim on the stiffener's start web
<USER>	The user name of the user to create the plot

SC Profile Plot Template > Insert BOM Table

Button .....
Ribbon ..... Structure Production tab > BOM panel > Insert BOM Table
Menu ..... SC Profile Plot Template > Insert BOM Table
Toolbar..... None
Command ..... SCINSERTBOM
Permissions ... None
Procedure..... None

Inserts an empty BOM into the current Profile Plot. You must create a BOM definition in Manager before inserting a table.
See BOM Definitions Manager (page 47).

SC Profile Plot Template > Replace Legacy Keywords

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Sheet Template panel > Replace Legacy Keywords
Menu ..... SC Profile Plot Template > Replace Legacy Keywords
Toolbar..... None
Command ..... SCPROFILEPLOTTEXTTOKEYWORD
Permissions ... None
Procedure..... None

To help update ShipConstructor2005 templates so they can be used in ShipConstructor this command replaces any known ShipConstructor2005 keywords in a template with the corresponding ShipConstructor keywords.

SC Profile Sheet Template Menu

The SC Profile Sheet Template menu appears within profile sheet template drawings (see Set Up Profile Plot Templates (page 296)).

SC Profile Sheet Template > Draw Border

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Sheet Template panel > Draw Border
Menu ..... SC Profile Sheet Template > Draw Border
Toolbar..... None
Command ..... SCPROFILEPLOTSETSHEET
Permissions ... None
Procedure..... None

Creates a new border for the profile plot sheet. If one exists in the drawing already it is removed. You can resize existing borders by moving the grip points.

SC Profile Sheet Template > Draw Area

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Sheet Template panel > Draw Area
Menu ..... SC Profile Sheet Template > Draw Area
Toolbar..... None
Command ..... SCPROFILEPLOTSETAREA
Permissions ... None
Procedure..... None

Creates a plot area for the profile plot sheet. If one exists in the drawing already it is removed. You can resize existing areas by moving the grip points.

SC Profile Sheet Template > Insert Keywords

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Sheet Template panel > Insert Keywords
Menu ..... SC Profile Sheet Template > Insert Keywords
Toolbar..... None
Command ..... SCINSERTKEYWORD
Permissions ... None
Procedure..... None

Keyword	Description
<COMPANY_NAME>	The value from Project Settings
<DATE_TIME>	The date and time the plot was created or last updated
<DATE>	The date the plot was created or last updated
<ENDCUT_END>	The endcut at the end of the stiffener
<ENDCUT_START>	The endcut at the start of the stiffener
<FILE NAME>	The file name only
<FILE>	The complete path and file name
<JOB NUMBER>	The value from Project Settings >
<MATERIAL>	The material and grade of stock for the parts in the plot
<PLOT_UNIT>	The common unit of the parts in the plot. If the parts belong to different units then 'Varies' will be displayed
<PROJECT DESCRIPTION>	The value from Project Settings >
<PROJECT NAME>	The value from Project Settings >
<SCALE>	The scale of the plot relative to the full size stiffener
<STOCK_DESC>	The Description field of the stock for the stiffeners in the plot
<STOCK_NAME>	The stock name of the stiffeners in the plot
<USER>	The user name of the user to create the plot

SC Profile Sheet Template > Replace Legacy Keywords

Button .....
Ribbon ..... Nest and Profile Plots tab > Profile Sheet Template panel > Replace Legacy Keywords
Menu ..... SC Profile Sheet Template > Replace Legacy Keywords
Toolbar..... None
Command ..... SCPROFILESHEETTEXTTOKEYWORD
Permissions ... None
Procedure..... None

To help update ShipConstructor2005 templates so they can be used in ShipConstructor this command replaces any known ShipConstructor2005 keywords in a template with the corresponding ShipConstructor keywords.

Index

3

3D unit drawings 151
3D, 2D view 24

A

accessory packages 336
angle
  dihedral 206
approval drawings
  bound 217
  create 216
  create template 215
  insert planar group 216
AutoCAD 378
  and curved plates 189
  and faceplates 178
  and stiffeners 167
  drawing 104
  drawing tool 75
  plot command 294
  trim 158
  UCS 136
  undo command 140
  xclip 16, 18

B

bevel information 166, 167
bevel standards 85, 357
bill of materials See BOM
BOM
  nest 255
  update nest 293
BOM Manager 47
bridges 282, 525
  delete 285, 525
  insert 284
  update 526

C

catamaran hulls, mirroring 195
CG 491
Clip view 455
composite drawings 16
construction lines
  management 150
  planargroupplane 393
  relationships 154
corner treatments 81, 161, 355, 366, 407, 408
  create 81, 83, 95
  delete 82
corrugated plate
  create 180
  edit 184
corrugated plates 180, 365, 423
  edit 365, 425
  properties 424
curved
  create 184
curved plate
  edit 188
  production information 189
curved plates 184, 189, 426, 428, See curved plates
  and hull drawing 186
  edit 427
  properties 426
cutout 67
  add to stiffener 171
  edit 172
cutout type 66
cutouts 162, 406
  and faceplates 179
  edit window 417
cutting paths 292

D

database 13, 151
Database Driven Relational Object Model See DDROM
datum lines 410
DDROM 3, 147
  best practices 148
  example 149
  relationship management 148
detailed report 314
dihedral angle
  determine 206
dimension
  insert to structural group 205
dimension styles 45, 339
drawings 14
  close 16
  composite 16
  create 15
  delete 16
  export 20, 22
  insert 17
  link 17

lock unlock 15
open 14
rename 15
save 16

E

edit sizes window 343
editors 24
endcut
definitions window 360
endcuts 92, 358
create 92
delete 94
export 94
formula 358
import 94

F

faceplate
bend tolerance 535
create 178
cutouts 179
edit 179
moldline 179
moving 179
trimming 179
faceplates 366, 420
add cutout 179
cutout 422
edit 366, 421
mold line 423
neutral axis 423
properties 420
trim 422
finish
create 33
create finish type 33
delete 34
export 34
import 34
rename 34
finishes 334
window 334
flange
catalog 79
create standard 79
edit 80
export 80
import 80
library window 352
flanges 163, 401
add 163
edit 402
edit plate 163
remove from plate 164

G

general use tips 151
getting started 7
grades
create 30
delete 31
export 30
import 30
rename 30
green 164, 361, 403
edit 403
green standards 355
group intersections 157

H

hull trace 442
hulls, mirroring (catamaran) 195

I

import hull model 107
insert profile plots 529
insulation 335
interferences 200, 510
check 200
create drawing 200
inventory
window 344
inverse bending 535

K

keywords
insert 540

L

label
insert to structural group 205
label styles 45
layer
activate 140
copy objects from layer 141
deactivate 141
move objects from layer 141
layers
activate 519
NC Pyros 519
visibility 518
licenses 12
licensing window 390

M

manager 3, 383
manufacturer, create 32

manufacturers 333
  window 333
mark group intersections window 393
mark symbols
  window 350
markline styles 73, 348
  assign 77
  create 73
markline styles window 73
markline symbols 74
  create 75
  delete 76
  edit 75
marklines
  assign markline styles window 351
  style window 348
materials 332
  create 29
  delete 31
  export 30
  import 30
  rename 30
  window 332
menus
  manager 327
  SC approval 488
  SC assembly 489
  SC assembly template 503
  SC interference 509
  SC profile plot template 539
  SC profile plots 529
  SC structure 393
  ShipConstructor 368
  structure 342
Menus
  SC Utilities 451
mirroring catamaran hulls 195
mlink 17
Mlink 383
Mlink manager 383
model link See mlink
model space 24
model structure 147
move and rotate 515

N

naming conventions 35, 336
  activate 40
  create 38
  delete 40
  edit element 39
  element properties 39
  remove element 39
  rename 40
  re-order elements 39
  window 336, 337
navigator 6, 368, 369
NC Code
  generate 292
  nests 290
NC Cutting
  hide, show paths 280
NC machines 90
  create 90
  delete 90
  edit 90
  export 91
  import 91
  window 357
NC Machines 356
  for nesting 255
NC tapes 294
NC-Machine
  assign 284
NC-Pyros 523, 525
  export window 523
  nest 290
nest 511
  assign NC Machine 284
  assign parts 273, 274
  BOM format 255
  check 521
  check and fix 288
  check for errors 288
  collision 289, 522
  cutting path 292
  delete 281, 512
  drawing 259, 260
  edit 513
  edit stock plate 281
  find 281, 512
  hide, show parts 280
  insert parts 262
  insert to drawing 261
  keywords 281
  list parts 277
  move or rotate parts 277
  naming convention 255
  NC code 290, 292
  NC Machine 513
  NC Pyros 290
  options and colours 254
  plot 294, 513
  re-insert parts 290
  remove parts 272
  re-nest 272, 512
  report 294
  select parts for nesting window 262
  template border 257
  template keywords 258
  templates 255, 256, 257
  update 522
  update BOM 293
  update keywords, BOMs 289
nesting 254
  NC Machines 255
  profile 305
  stocks 255

O

object enabler 23
objects

convert 3D to 2D 207
display in mlink 142
display wireframe, hidden line, shaded mode 142
duplicate to other planar group 203, 204
hide or show 141
open a project 11
orientation icon 72, 346, 490, 504
3D/2D window 490, 504
window 346

P

paper space 24
parametric modeling 3
parts
add manually to profile nest 309
add object 160
add, remove from profile nest 309
align 516
assign to nest 273, 274
automatic profile nest 310
change, check profile nest 311
corrugated plates See corrugated plates
create related 155
display related 156
faceplate See twisted stiffeners
hide, show 280
info 433
insert 514
into nest 262
list 277, 517
list un-nested 278
mirror 195
mirror about centerline 195
move or roatate 277
move or rotate 515
place along edge 279
planks See planks
plate See plate parts
re-insert 290, 515
relationships 155
remove 515
remove from nest 272
remove from profile nest 311
rotate 516
slide along edge 275, 515
snap 517
standard 193, 362
stiffener See stiffeners
twisted stiffeners See twisted stiffeners
un-nested 518
update 490, 509, 510
password 13
piecemark
style window 345
piecemark styles 70, 344, 345
activate 71
create 70
edit 71
options 71
piecemarks 195
change position 196
change size 195
planar group
duplicate objects 203, 204
planar group drawing
create 127
create from 2D structural drawing 131
create template 127
transfer objects 133
transfer objects from unit drawing 134
planar groups 127, 441
creation 155
delete 133
edit properties 133
transfer 133, 134, 384
planks 189, 428
create 190
delete 193
edit 430
grip points 192
properties 429
plate green 83
plate parts 158
create 158
plates 365, 397, 413, 414
edit 365, 400
insert 132
polyline
convert ellipse or spline 208
polylines
reduce number of vertices 207
product hierarchy
check 211
production information 189
profile
nesting 305
select assembly for nesting 305
profile green 95
profile nest
create empty 307
create or delete 307
delete 308
generate report 313
issue 312
issue, un-issue 312
manager window 387
set to cut 315
set to uncut 316
un-issue 313
profile nesting 386
profile nests
add parts manually 309
add, remove parts 309
automatically create 310
change, check parts 311
remove parts 311
profile plot
check 304
create drawing 299
create template 298
define area 298
define border 297, 298
insert into drawing 301

insert keywords 298, 299
output 534
set up template 296
setup window 532
template menus 539
zoom 303, 304
profile plot drawing
insert profile plot 301
zoom 303, 304
profile plots 295
check 538
options 532
print 304, 537, 539
set up 300
zoom 536
profile stock
create shape 54
edit shape 66
project database 13
project settings 27, 327
export 28
import 28
set 27

Q

quality matrix 462
options 463

R

remnant
delete 308
remnants 285
create empty 307
create or delete 307
delete 288, 514
Remove clip 455, 456
report
generate summary or detail 314
stock use 315
rofile
set up stock 305

S

shape properties
angle stock 54
bulb flat stock 56
channel stock 56
custom stock 65
flat bar stock 57
round bar stock 59
structural pipe stock 60
tee stock 61, 62
W stock 63, 64
shapes
project 207
ShipCAM 175, 468
slide function 276, 515
standard parts 97, 362
create 98
edit 99
rename 100
set up 97
stiffener
attach to plate part 174
create 167
cutouts 171
edit 169
mold line 174
moving 174
trimming 170
stiffeners 167, 366, 412, 414
add cutout 171
cutout 417
edit 366, 414
neutral axis 419
plot options window 532
polyline 420
properties 412, 530, 531
start and end position 167
trim 366, 418
twisted 415
twisted, properties 415
weld seam reliefs 419
stock
properties 343, 402
stock plate, edit 281
stock properties
corrugated plate 52
plate 52
profile 53
rect tube 58
stock usage report 315
stocks
catalog 51
create 51
create profile shape 54
edit 53
edit profile 66
export 69
for nesting 255
import 69
plate stock properties 52
view use 69
structural display options window 394
structural modeling 151
structural stock editor window 342
style to copy 533
summary report 314
system font 7

T

templates 104
optional 104
required 104
text styles 35, 335
create 35
delete 35
toolbar

assembly 477
equipment 363
hull 363
hull curve 363
hull expand 363
hull pin jig 363
hull porcupine 363
hull stringer 364
hull surface 364
labels 364
nest 478
penetrations 364
ShipConstructor 364
structural tools 364
structure 364
structure display 367
utilities 367
toolpath 276
toolpaths 205
create closed 206
trim
window 418
twisted stiffener
clean geometry 177
cutouts 177
edit 176
moldline 177
moving 177
trimming 177
twisted stiffeners 175, 178
clean geometry 416
edit 416
properties 415

U

UCS 451
activate 138
create out of plane 136
create parallel 137
select viewpoint 139
unit
create 107
delete 108
setup 107
unit drawing 134, 211
user coordinate systems (UCS) 136
user permissions 332
user-defined attributes 49, 345
assign 50
create 49

V

viewport 493, 506
views 24

W

weld reliefs 173
weld seam reliefs 419
weld shrinkage icon 165, 410
window 410
windows 24
work package 294

X

XML file 327